NASA Technical Memorandum 106542
ICOMP-94-01

IN-34
9923
91P

Institute for Computational Mechanics in Propulsion (ICOMP)

Eighth Annual Report – 1993

(NASA-TM-106542) INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP) Annual Report No. 8, 1993 (NASA. Lewis Research Center) 91 p

N94-32969

Unclas

G3/34 0009923

April 1994

National Aeronautics and
Space Administration

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION
NASA
ICOMP
Lewis Research Center
Ohio Aerospace Institute

NASA Technical Memorandum 106542
ICOMP-94-01

Institute for Computational Mechanics in Propulsion (ICOMP)

Eighth Annual Report—1993

Compiled and edited by
Dr. Charles E. Feiler
ICOMP Executive Officer

Approved by
Dr. Louis A. Povinelli
ICOMP Director

April 1994

CONTENTS

INSTITUTE FOR COMPUTATIONAL MECHANICS

IN PROPULSION (ICOMP)

EIGHTH ANNUAL REPORT

1993

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by the Ohio Aerospace Institute (OAI) and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the accomplishments and activities at ICOMP during 1993.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in September 1985, to improve problem-solving capabilities in all aspects of computational mechanics relating to propulsion. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion.

The scope of the ICOMP program is: to advance the understanding of aerospace propulsion physical phenomena; to improve computer simulation of aerospace propulsion components; and to focus interdisciplinary computational research efforts. The specific areas of interest in computational research include: fluid mechanics for internal flows; structural mechanics and dynamics; and fluid-structural interactions. In 1990 a specific new focus was added with the formation within ICOMP of the Center for Modeling Turbulence and Transition (CMOTT).

The ICOMP program is administered by the Ohio Aerospace Institute (OAI) under NASA Cooperative Agreement NCC3-233. Since January 1993, most of the ICOMP research staff has been located in the OAI building, a very happy arrangement. The daily management of ICOMP is provided by Program Director, Dr. Louis A. Povinelli, OAI Director for Workforce Enhancement, Dr. Theo G. Keith, Jr., ICOMP Executive Officer, Dr. Charles E. Feiler, and Program Manager, Ms. Karen L. Balog.

The remainder of this report summarizes the activities at ICOMP during 1993. It lists the resident and visiting researchers, their affiliations and educational backgrounds, followed by reports of RESEARCH IN PROGRESS, REPORTS AND ABSTRACTS published and SEMINARS presented. There were several other special activities during 1993. One was the sponsoring of a series of seminars focusing on Computational Aeroacoustics. These seminars were organized by Dr. Reda Mankbadi of the Lewis

Research Academy and are denoted in the SEMINARS section by a double asterisk (**). A second was the sponsoring of a two-day workshop on Computational Turbulence Modeling held at NASA Lewis on September 15-16, 1993 for Lewis and Lewis-affiliated personnel. This workshop, organized by the CMOTT group under the technical leadership of Dr. T.H. Shih, was a part of the continuing effort to introduce improved turbulence and transition models into the CFD community. An overview summary of this workshop is given.

Finally, in a separate activity designed to evaluate and direct the overall LeRC turbulence modeling and transition research programs, the programs were subjected to a two-day peer review held at LeRC on September 20-21, 1993. This review involved both CMOTT and LeRC research efforts including those at the Lewis Research Academy. The members of the peer review committee were: T. Coakley, NASA Ames; J.Y. Chen, University of California,Berkeley; J.L. Lumley, Cornell University; N.N. Mansour, NASA Ames; C. Prakash, General Electric Co.; M. Sindir, Rocketdyne; and S. Syed, Pratt& Whitney Aircraft Co. The recommendations and conclusions from this review are being digested at present.

THE ICOMP STAFF OF VISITING RESEARCHERS

The ICOMP research staff for 1993 is shown in Table I. A total of sixty-four researchers were in residence at Lewis for periods varying from a few days to a year. The resident staff numbered twenty-six, while the visiting staff, including three graduate students,numbered thirty-eight. Table II shows the growth of ICOMP during its first six years in terms of staff size and technical output as measured by the numbers of seminars, reports and workshops. Figure 1 is a photograph of the ICOMP Steering Committee and the visiting researchers taken at a reception in July 1993. The photograph gives a glimpse of the spectacular and exciting new OAI building that houses most of the ICOMP staff. The next sections will describe the technical activities of the visiting researchers starting with reports of RESEARCH IN PROGRESS, followed by REPORTS AND ABSTRACTS, then SEMINARS, and finally, the workshop on COMPUTATIONAL TURBULENCE MODELING.

RESEARCH IN PROGRESS

D. AFOLABI

During my residence at ICOMP in 1993, I continued previous work on the problems of flutter and other forms of mechanical and aeroelastic instability in propulsion systems. I extended prior work on the application of recent mathematical techniques—such as transversality theory, catastrophe theory, and bifurcation theory—to the solution of instability problems in propulsion systems. Two ICOMP Reports [1,2] initiated during the previous ICOMP visit were completed this summer. The weak transversality theorem was used in the two papers. Catastrophe theoretic techniques, which I used in a related but different paper [3], were also used in [1] and [2]. In all these Reports, new and significant results were obtained. A third ICOMP Report [4] is currently in preparation. This report is designed for interpretation of experimental data on rotating blades, disks, bladed-disks, etc., and complements on-going experimental work at the Structural Dynamics Branch of NASA Lewis Research Center.

References

[1] Afolabi, D.: Flutter Analysis Using Transversality Theory. ICOMP Report in Preparation. (Accepted for publication in Acta Mechanica).

[2] Afolabi, D., and Mehmed, O.: On Curve Veering and Flutter of Rotating Blades. ICOMP Report 93-27, NASA TM 106282, August, 1993. (Accepted for publication in J. Turbomachinery).

[3] Afolabi, D.: The Cusp Catastrophe and the Stability Problem of Helicopter Ground Resonance. Proc. R. Soc. Lond., ser. A, vol. 441, pp. 339-406 (1993).

[4] Afolabi, D.; Lucero, J.; and Mehmed, O.: Transformation of Experimental Data from Rotating Frame to Stationary Frame. ICOMP Report in preparation.

KUMUD AJMANI

The High Speed Civil Transport (HSCT) objectives of the High Performance Computing and Communications (HPCC) program depend heavily on the design of methodologies for multi-disciplinary analysis and design. Massively parallel implementations of such design methodologies would be required to ensure improved computational efficiency and tractability for large problems. The current research being conducted at ICOMP is strongly motivated by this need for the development of parallel, multi-disciplinary design techniques.

Computational Fluid Dynamics (CFD) has matured sufficiently to the point where flow solutions provided by advanced CFD codes can be reliably used to extract information for use in aerospace vehicle design. A typical CFD code solves a system of partial differential equations on a discrete mesh, for a given, fixed, set of flow conditions (like Mach number, Reynolds' number etc.). However, practical design codes usually require lots of additional information in the form of sensitivity derivatives, in order to produce an optimal design.

The current work focuses on obtaining such sensitivity derivatives by solving an 'incremental' form of the sensitivity equations derived by differentiating the discretized thin-layer Navier Stokes equations with respect to certain design variables of interest. The equations are solved with a parallel, preconditioned Generalized Minimal RESidual (GMRES) solver on a distributed-memory architecture.

The 'serial' sensitivity analysis code is parallelized by using the Single Program Multiple Data (SPMD) programming model, domain-decomposition techniques, and message-passing tools. Sensitivity derivatives are computed for flow over a NACA 1406 airfoil on a 32-processor Intel Hypercube, and found to be identical to those computed on a single-processor Cray Y-MP. The parallel GMRES solver provides consistent and accurate sensitivity derivatives for both low Reynolds number (laminar) and high Reynolds number (turbulent) flows. The accuracy of the computed sensitivity derivatives is found to be independent of the number of processors. This research indicates that a finite-difference method of calculating sensitivity derivatives is more accurate than a quasi-analytical method, particularly for high Reynolds number (turbulent) flows.

The quasi-analytical method of calculating sensitivity derivatives is 25% more efficient than the finite-difference method, in terms of processing time. The parallel processing times for both the low and high Reynolds number test cases indicate that 40-50 parallel processors of an Intel Hypercube would match the performance of a Cray Y-MP. The parallel, preconditioned GMRES solver exhibits similar processing time characteristics and scalability when calculating sensitivity derivatives for both the laminar and turbulent flow cases.

In future work, the procedure for obtaining sensitivity derivatives developed in this research will be tested on larger parallel machines. This will be done to further study the scalability of the code, and the effectiveness of the parallel solver on large numbers of processors. The sensitivity analysis code will also be ported to a 'cluster' of workstations at NASA Lewis, in order to study its performance characteristics in a loosely coupled parallel environment.

ANDREA ARNONE

During my visit at ICOMP, I continued work on the TRAF (2D/3D) codes. These two- and three-dimensional viscous solvers were developed during a joint project between the University of Florence and NASA Lewis and were designed for turbomachinery blade row analysis. Presently, activities are carried out on mainly three topics:

1. Accounting for unsteadiness; a three-dimensional unsteady multi-grid release of the TRAF3D code is near completion.

2. Study of the role of leading edge flow in transonic compressors; a grid sensitivity analysis has been carried out on the NASA Rotor 67 with particular emphasis on leading edge flow resolution. The results of this study will be beneficial to the Rotor 37 blind test case.

3. Extension of the TRAF code to handle multiple blade rows; a mixing plane method has been implemented and is under investigation.

During my summer visit, I also continued my cooperation with the Turbomachinery Fluid Physics Branch where TRAF2D and TRAF3D codes are employed for calculations in the area of blade passage heat transfer, film cooling, and estimation of losses.

THOMAS F. BALSA

Much work has been done during the past few years by M. E. Goldstein and his colleagues at the NASA Lewis Research Center on the evolution of nonlinear spatial instability modes in a variety of shear flows (e.g. subsonic and supersonic mixing layers and boundary layers). These studies employ the concept of "nonlinear critical layers" in order to calculate the amplitudes of the modes. Our work this summer focused on extending these ideas to spatio-temporal disturbances (i.e. modulated disturbances or wave packets) in incompressible mixing layers and boundary layers.

For the case of the mixing layer, a narrow band (say of $O(\sigma) << 1$) of wavenumbers and frequencies in the vicinity of the neutral point is included in the analysis. The amplitude of the wave packet scales as σ^2 in order to obtain a strongly nonlinear critical layer. This amplitude and the vorticity in the critical layer obey a coupled system of nonlinear partial differential equations. In the limit of very small amplitudes (i.e. in linear theory) our equations reduce to that of Benney & Maslowe, and for spatial instability modes we recover the results of Goldstein & Hultgren.

For the case of the boundary layer in a small adverse pressure gradient (say of $O(\sigma^2) << 1$), the problem is considerably richer. The instabilities in this flow are confined to wavenumbers of $O(\sigma)$ with growth rates of $O(\sigma^4)$. Wave dispersion occurs at lower orders; therefore, it is possible to choose a family of scalings which reflect different balances between the dispersion and the instability of the wave packet. This family is generated by allowing the bandwidth to depend on different powers of σ. Work is continuing along these lines and to include the effects of three-dimensionality.

GILES J. BRERETON

During my visit to ICOMP, I completed a review article on progress in the understanding and prediction of unsteady turbulent flows in collaboration with Dr. R. Mankbadi. I also worked on developing the rapid-distortion approach to modeling unsteady flows, with particular emphasis on modeling unsteady dissipation of turbulent kinetic energy. This work is of relevance to forced and naturally unsteady flows, turbulent/acoustic interactions, and dynamic sub-grid-scale modeling for large eddy simulations of turbulence. I have also used it as a basis for proposing a distortion-strain lengthscale which characterizes numerous unsteady-flow phenomena such as streak spacing. The present direction of the modeling research is to extend the results presented at the International Conference on Near-Wall Turbulent Flows, Tempe, March 1993 (ICOMP 93-22 and NASA TM 106249) to include a more precise treatment of the near-wall region and to allow a better description of the relaxation to equilibrium behavior. This work will continue throughout the year and should result in publications shortly.

During my visit, I had numerous fruitful discussions with Dr. Mankbadi. I also had the opportunity to interact with Jim Holdeman, Z. Yang, and numerous others.

J.-Y. CHEN

The main objective of this research is to study the interaction between turbulence and realistic chemical kinetics in nonpremixed turbulent flows using Direct Numerical Simulations (DNS). The starting point is the development of a feasible strategy of carrying out realistic chemical kinetics in DNS without adding a significant burden to the computational task. Such a task can be accomplished by considering specially designed fuel mixtures of H_2/AR and in a specific regime of turbulent combustion.

By properly choosing the contents of the fuel mixtures, both the flame thickness and the degree of stiffness caused by chemical kinetics can be adjusted so that the flame will be fully resolved both in time and space. The computational task needed for the realistic chemical kinetics can be made manageable by using reduced chemical reaction mechanisms and efficient algorithms for evaluating chemical kinetics source terms.

The present development includes: (1) construction of a look-up table for the source terms for the hydrogen chemistry, $2H_2+O_2 \Rightarrow 2H_2O$ and for the Zeldovich NO mechanism, $N_2+O_2 \Rightarrow 2NO$; (2) implementation of the interpolation scheme into a high-order finite difference DNS code for compressible homogeneous turbulence as well as a spectral code for incompressible flows; and (3) testing of these newly developed computer codes. This development facilitates studies of the effects of density variation, compressibility, and the interaction of chemistry on turbulence. Studies have been conducted for homogeneous turbulence with and without a forcing scheme to maintain kinetic energy. The preliminary results reveal that the compressibility causes the flame temperatures in either lean or rich sides to be higher than the counterparts in incompressible flows. A significant departure from chemistry equilibrium is also observed as the turbulence time scale can be smaller than the characteristic chemistry time scale. Future development is planned to impose a constant shear on the homogeneous turbulence.

TAWIT CHITSOMBOON

The mixer-ejector concept has been proposed to reduce the noise level of the High Speed Civil Transport (HSCT) aircraft, by reducing its exhaust jet velocity through mixing the hot jet exhaust with the slower entrained ambient air. A 2D, rather than axisymetric, nozzle configuration has thus been proposed to achieve this objective. The 2D nozzle is partially enclosed by a shroud/ejector system in order to entrain ambient air mass as well as to absorb the noise. This is an extremely complex fluid flow device to analyse. Modern CFD is very applicable for this type of problem and computational effort to understand the flowfield inside the nozzle of the HSCT has been continued from the previous year.

During the past year, several runs were made using the MAWLUS code with a high Reynolds number κ-ε turbulence model. The computational results compared quite reasonably with the experimental data of Pratt-Whitney; but, in general, MAWLUS always underpredicted the entrainment rate of the ambient air.

The NPARC code has also been used. It was found, quite paradoxically, that a coarse-grid produced better solution than a fine-grid; and that the values of artificial viscosity coefficients could somewhat change the quantitative behavior of the solution. With the artificial viscosity coefficients set at the theoretically correct values, however, NPARC, like MAWLUS, also underpredicted the entrainment rate.

It should be noted that this is a turbulent shear-driven flow. The above conclusion for NPARC was drawn based upon the use of an algebraic turbulence model which may not be applicable for this very complex flowfield. The next attempt was to try the κ-ε model. Unfortunately, as had been found by some other investigators, the κ-ε model failed to work. Some effort was spent to find the code errors and to reset minimum values of κ and ε in a consistent manner. With these fixes, NPARC with the κ-ε model worked just fine, but, the entrainment rate was again underpredicted. The next step was to look into the turbulence model itself. It was found by Nick Georgiadis of the Nozzle Technology Branch that Speziale's model which was used in NPARC produced inferior results as compared to Chien's model for a similar type of flowfield in a two-dimensional ejector problem. It was concluded then that Speziale's model in NPARC should be converted into Chien's model. The code alteration to convert to Chien's model has been completed and the code is now being tested.

JOONGKEE CHUNG

Based on the evaluation of unsteady PARC 2D code, turbulent flow calculations for engine inlets were performed for realistic simulations. To adequately deal with the formation of large separation regions, a bleed hole was placed upstream of the throat, which eliminated some separation. Placing a bleed slot instead of using a mass flow boundary condition directly at the bleed hole resulted in better mass flow calculation.

One of the concerns of the engine inlet numerical computation is the relation with the controls. The engine often encounters various upstream and downstream perturbations. Thus, the diffuser exit, upstream, and bleed boundary conditions were modified and improved to satisfy the requirements of engine controls. Also, the capability of real time display for contours of flow properties like density, pressure, and Mach number was incorporated into the PARC code. Using this capability, various properties at specific field points can be monitored in real time while the computations are being performed on any computer.

To further extend the unsteady numerical simulation, a three-dimensional engine inlet geometry modeling was done by means of ICEM CFD software which has grid generation capabilities as well as CAD capabilities. The interactive nature of the ICEM CFD allows the direct decision-making necessary for complex and unique geometries. The approach allows the geometry to be modeled in the most accurate manner independent of the grid generation. The complexity of the geometry required a multi-block approach for generating the structured grid.

Several multi-block three-dimensional grids were constructed for the computation of the unsteady, inviscid flow in a Variable Diameter Centerbody (VDC) inlet. A 3D time accurate version of PARC code was used for the prediction of flow properties as well as the location of the moving normal shock with respect to time. Due to the existence of struts in the diffuser section, the flow properties were not uniform in radial direction. Various angle of attack simulations were also performed but due to the lack of experimental data, comparisons between numerical and experimental results could not be made.

FREDERIC COQUEL

I worked in collaboration with Dr. Meng-Sing Liou of the NASA Lewis Research Center on a mathematical formulation of a hybrid upwind splitting scheme. The HUS strategy yields a family of upwind methods that exhibit the robustness of Flux Vector Splitting (FVS) schemes in the capture of nonlinear waves and the accuracy of some Flux Difference Splitting (FDS) in the resolution of linear waves. This research is in its final stages and a paper describing it will be completed soon.

THONG DANG

Most of my short visit to ICOMP was used to introduce a newly developed 3D inverse method for the design of turbomachinery blades. A summary of this method is given in the abstract by Thong Dang which appears in the SEMINARS section of this report. One of the main features of this new method is its ability to design supersonic throughflow fan blades.

DOMINIC DAVIS

Together with Professor Art Messiter, of the University of Michigan, and Dr. S. J. Leib, of NASA Lewis Research Center, I have been considering the nonlinear spatial evolution of a pair of initially-linear instability waves in two-dimensional (2D) free shear layers. The waves are slightly non-planar in the sense that there is a small, but non-zero, angle of wave propagation in the streamwise-spanwise plane. The research builds on the work of Goldstein and Hultgren (1988) for strictly 2D waves. As in their paper, the flow structure consists of an inviscid shear layer surrounding a thinner non-equilibrium critical layer. Inside the critical layer, there are significant nonlinear and viscous effects both of which are absent outside. The small angle of wave propagation implies the existence of a new vortex stretching term in the critical-layer spanwise vorticity equation, which is absent for strictly 2D waves. Computational solutions are presently being sought based on a code provided by Dr. David Wundrow.

Reference

[1] Goldstein, M. E. & Hultgren, L., 1988. Nonlinear spatial evolution of an externally excited instability wave in a free shear layer. J. Fluid Mech., 197, 295-330.

ROBERT DEISSLER

Work on Marangoni convection and instability was continued in collaboration with Dr. Alexander Oron of ICOMP and Dr. J. C. Duh of Sverdrup. A paper on this subject has been published (Deissler, Robert J.; Oron, Alexander; and Duh, J. C.: "Marangoni Instability in a Liquid Layer With Two Free Surfaces," ICOMP Report 93-15, NASA TM-106166, July, 1993). Some further details of this research are given in this section in the report by Alexander Oron.

A second project, done in collaboration with Dr. Wai-Ming To of Sverdrup, deals with the convective amplification of noise and the resulting noise-sustained structure for Taylor-Couette flow with an axial through-flow. A paper on this work has also been published (Deissler, Robert J.: "Thermally-Sustained Structure in Convectively Unstable Systems," ICOMP Report No. 93-39, NASA TM 106375, November, 1993).

PETER DUCK

I have been studying a number of so-called three-dimensional (inviscid) stability problems. This study involves the investigation of the stability of base-flows which are dependent on two spatial variables. A small amplitude perturbation (periodic in the third spatial coordinate) is then imposed on this flow, and the temporal behavior of this perturbation is then analyzed. In particular, I considered the large Reynolds number limit of this problem, which then permits the use of the 'three-dimensional' Rayleigh equation. This is a two-dimensional partial-differential equation, involving an unknown wavespeed/growth parameter, which must be determined in order to decide the stability (or otherwise) of the flow. This has been attempted using a variety of different computational techniques. The first involves a direct computation of the wavespeeds, using (after spatial discretisation) the QZ algorithm. This has the advantage of producing many (in fact N, where N is the total number of grid points) eigen-solutions. However, the computer storage required for this technique is $O(N^2)$, which rapidly becomes significant (together with the required c.p.u. time). A second technique involves a so-called local search, in which a wavespeed is guessed, and then iterated upon in order to satisfy the same equations as used in the above, first scheme. A third approach, which at the time of writing seems to be the superior method, involves taking the base flow, introducing a small amplitude distortion to this flow, and time-marching the solution. This has the advantage that no *a priori* knowledge of the wavespeed is required, and the computer storage requirements are relatively modest. This scheme, although only producing one eigensolution, does yield the eigensolution with the largest growth-rate, *i.e.* the most important mode. Although the stability of a variety of flows can (and has been) analyzed, most attention has been focused on the oscillatory flow inside a rectangular cavity.

Additional work (in conjunction with Dr. Leib) has been to study the so-called "inviscid triple-deck equations." This problem arises from the breakdown of the critical layer structure, arising from resonant-triad interactions in an adverse-pressure-gradient boundary layer (see Goldstein & Lee, 1992, J. Fluid Mech. **245**, 523). Specifically, the problem involves the solution of a lower-deck region (governed by the unsteady Euler equations, but with zero normal pressure gradient), simultaneously with the solution of an upper-deck region, where the pressure field is harmonic. A numerical scheme, based on that of Bodonyi, Welch, Duck & Tadjfar, (1989, J. Fluid Mech. **209**, 285) is currently being implemented to study the problem.

SCOTT DUDEK

My research for the summer of 1993 worked toward using an Adaptive Mesh Refinement (AMR) algorithm, similar to that used by Berger and Colella for inviscid compressible flow, to solve the steady, compressible, Navier-Stokes equations in complex geometries using structured, mapped grids. In pursuit of this, work this summer included writing a hybrid C++/Fortran code to solve the Poisson equation (using multigrid iteration) on mapped grids which contain multiple levels of refinement. All of the high-level data structures associated with AMR are expressed in the object-oriented language C++, which also allows simple memory management and recursion, while the Fortran portion of the code, which vectorizes more efficiently, handles the floating-point operations. In addition, the code uses already-existing C++ libraries, developed at the Lawrence Livermore National Laboratory for general use in adaptive algorithms. The present code will be extended in the fall of 1993 at UC Berkeley to solve the steady, compressible Navier-Stokes equations using the multistage Runge-Kutta method with upwinding developed by Arnone, Liou, and Povinelli at NASA Lewis.

This research is part of a larger collaboration between myself and Phil Colella at UC Berkeley, and Erlendur Steinthorsson and Dave Modiano at ICOMP, to develop adaptive mesh codes, both time-accurate and steady, for compressible Navier-Stokes. These efforts will be closely coupled, and we will continue to collaborate throughout the next year.

JACK R. EDWARDS

The focal point of my work at ICOMP was the addition of full multigrid capability to an existing line/planar upwind relaxation solver for the 2-D/3-D compressible Navier-Stokes equations (AIAA J., Vol. 31, No. 1; AIAA Paper 93-0540). An initial two-grid effort, presented as AIAA Paper 93-3317, revealed that highly implicit upwind relaxation algorithms can smooth or expel a wide range of error frequencies in an efficient, stable manner, making them well-suited for incorporation within a multigrid context. A more rigorous extension to multiple grids, accomplished using a combination of FAS and Newton-multigrid ideas, substantiated the earlier conclusions, providing four-fold to eight-fold improvement in time to convergence for several 2-D/3-D shock-separated internal flow computations

(15,000 - 375,000 grid points). As a further extension, the hybrid upwind - central differencing strategy originally incorporated in the relaxation solver was replaced by a full upwind scheme based on the AUSM approach of Liou and Steffen (NASA TM-104404). An exact, primitive-variable linearization of the AUSM scheme was developed for use within the implicit operator. Preliminary testing using the multigrid approach indicated that the AUSM discretization converged as well or better than the hybrid formulation.

Work also continued on the development of a diagonal implicit multigrid-based scheme for 2-D chemically-reacting, viscous flows. The AUSM scheme and several monotonicity-related modifications proposed by Y. Wada (ICOMP) and M. S. Liou (IFMD) were implemented and tested for conditions characteristic of atmospheric reentry. Good convergence rates were generally observed, although for extreme cases, fast reaction rates severely limited the allowable time step.

The third phase of my summer research at ICOMP involved the detailed evaluation of a nonequilibrium eddy viscosity - transport turbulence model currently under development at North Carolina A&T State University. The predictions from the model were compared with boundary layer profile and surface property data for two of the shock-separated, two-dimensional flows outlined in the Hypersonic Shock/Boundary Layer Interaction Database (NASA CR-177577).

J. S. B. GAJJAR

Transition in 3D boundary layers is fundamentally different from that in planar boundary layers because of the occurrence of cross-flow instability in the former. The combination of a streamwise velocity component with the spanwise component gives rise to a profile which can be inflexional in certain directions and hence subject to strong inviscid instability. Stationary cross-flow vortices occur due to the instability of a particular profile in which the effective flow velocity is zero at a point inflexion. The problem that I looked at during my visit this year was the nonlinear development of stationary cross-flow vortices in incompressible boundary layer flows. Unsteady nonlinear critical layer theory was used to derive a novel amplitude equation describing the slow spatial and temporal evolution of the stationary vortex. This amplitude equation takes the form of an integro-differential equation as in many critical layer studies, but the kernal function is different from one obtained previously. The properties of this equation are currently being studied. One other feature of the work worth mentioning is that the linear dispersion relation can be written analytically and the growth/decay of the wave is affected by the response of an unsteady wall layer. The linear dispersion relation is being studied analytically/numerically to ascertain whether this particular flow is absolutely or convectively unstable.

BERNARD GREENSPAN

Work is proceeding on the development of the Space-Time Conservation Element and Solution Element method originally proposed by Sin-Chung Chang. Three distinct projects are underway. The first is the application of the method to the solution of the energy equation for steady two dimensional channel flow. The second is a stability analysis of the method as applied to the one dimensional unsteady Navier-Stokes equations. The third is the analysis of the adaptation of the method to boundary value problems for ordinary differential equations.

MAX D. GUNZBURGER

The main thrust of our efforts has been towards the development, analysis, and implementation of finite element least-squares algorithms for partial differential equations, with special emphasis on the Navier-Stokes equations. This past year, we have had two major accomplishments. First, we undertook a careful computational study of the accuracy of the method, with particular emphasis on the role of boundary conditions. We found that the method provides very accurate approximations; and in fact, for the vorticity, we found that the method provides more accurate answers than any other method that uses the vorticity. This work was reported on in an ICOMP Report and has appeared in *Computers and Fluids.* Second, we have given the first rigorous analysis of the errors for the method. This analysis has resulted in an improved version of the method that is more accurate than previous methods. The improvement is to introduce appropriate mesh-dependent weights into the least squares functional. Although these weights are not necessary for the stability of the algorithm, they are necessary for optimal accuracy. Also, the introduction of weights represents a trivial coding change. Thus, the new improved method is easy to implement and gives the best possible accuracy. A report on this work is currently being prepared.

We have also been involved with flow optimization problems. We have finished a paper, which has appeared as an ICOMP Report and has been submitted to *Computer Methods in Science and Engineering*. This paper deals with the determination of optimal boundary heating and/or temperature controls that are necessary in order to get a uniform temperature on a desired portion of the boundary of the fluid. The fluid flow is coupled with adjacent solid bodies. Our work represents the first successful mathematical and computational treatment of coupled fluid/solid thermal control problems.

THOMAS HAGSTROM

1. Boundary Conditions for Unsteady Compressible Flow Simulations

In ongoing work with Professor S. I. Hariharan of the University of Akron and ICOMP, accurate radiation type boundary conditions are being developed and tested for unsteady compressible flow simulations in exterior domains. Accurate conditions for the wave equation are based on progressive wave expansions. We have successfully developed a procedure for computing such expansions for hyperbolic systems. The resulting boundary conditions, which we usually write in terms of local Riemann variables, have the desirable property of involving only time tangential derivatives. We have computed the coefficients in the expansions through second order for the Euler equations linearized about a uniform flow in both two and three dimensions. We have also computed the expansions to higher order for Maxwell's equations.

Boundary conditions based on these expansions have been implemented and tested in nonlinear, two-dimensional simulations. Our test problems all involve subsonic or transonic flow past a cylinder. We have considered both the case of an impulsive start and the case of a pulsating cylinder. In the future, vortical disturbances will be introduced, to simulate a "gust". In all cases, shocks must propagate through the artificial boundary. We have found that the second order conditions are both stable and more accurate than the usual procedure of setting incoming characteristics to their free stream values. Greater gains in accuracy are likely to occur for truly linear problems, as in aeroacoustics applications.

In joint work with Professor Jens Lorenz of the University of New Mexico, we have also derived boundary conditions for the linearized compressible Navier-Stokes equations at low Mach numbers. These will be used in simulations of problems involving sound wave - vortex interactions.

2. Development of a High-Order Numerical Algorithm for Reacting Flows in the Zero Mach Number Limit

In joint work with Dr. K. Radhakrishnan of Sverdrup, we have developed an experimental high-order numerical algorithm for simulating unsteady reacting flows governed by the zero Mach number asymptotic limit of the governing equations. Features of the code are:

(i.) High order spatial and temporal discretizations and adaptive meshing.

(ii.) The ability to incorporate general reaction and diffusive transport mechanisms.

(iii.) Solution of asymptotic equations which include a strong coupling between the reaction and the flow field, but which exclude the complicating feature of "fast" sound waves.

So far, we have only considered one-dimensional problems and simulated a few cases. We employ unsplit time marching schemes, based on BDF formulas, with a variety of implicit and explicit difference approximations for spatial derivatives. A key to the performance of the method is the iterative scheme used to solve the implicit system. Currently, we use a Newton-type algorithm on the chemical reaction terms combined with simplified diffusion approximations. These are accelerated by a general quasi-Newton technique. Future plans include experimentation with other iterative methods and variable step BDF formulas. We also plan to analyze our mesh adaptation scheme, which is based on explicit coordinate mappings.

S. I. HARIHARAN

During this year work on artificial boundary conditions was continued. This had several parts. Ongoing work with T. M. Hagstrom, The University of New Mexico, lead to a general theory for linear hyperbolic systems. This theory consists of the development of geometrical optics arguments for hyperbolic systems. This has many applications. Among them are the anisotropic linearized Euler equations and the isotropic Maxwell's equations. Preliminary tests were made on a transonic flow past a cylinder. This is a nonlinear problem. Results obtained indicate asymptotic improvement as the order of the conditions were increased. Linearized boundary conditions were used in the far field. Currently, to validate the theory, problems governed by the linearized Euler equations are being considered.

Work with M. E. Hayder (ICOMP), R. Mankbadi (NASA Lewis Research Center) and J. N. Scott (The Ohio State University) examines effective outflow boundary conditions for the jet noise problem. Several boundary conditions were tested including the one proposed by Hagstrom and Hariharan in 1988. This work is still in progress. A review paper on boundary conditions for this class of problems is under preparation with J. N. Scott.

Simultaneous efforts on testing the accuracy of various classes of boundary conditions for the Euler equations were carried out with D. K. Johnson at the University of Akron. These studies were made on a spherically symmetric flow problem. Generalizations to three-dimensional flows are under consideration with Hagstrom and D. Thompson (The University of Akron).

Earlier this year Hagstrom, Hariharan and R. C. MacCamy (Carnegie-Mellon University) developed a theory for the long time behavior of solutions for the two-dimensional wave equation. In this theory a nonconstant-coefficient local radiation boundary condition was proposed. Applications of this condition as an On Surface Radiation Condition for electromagnetic scattering problems has provided excellent results. The results will appear in an ICOMP report in the near future.

M. EHTESHAM HAYDER

In the past year, I collaborated with Dr. Reda Mankbadi (NASA Lewis) and several visitors at ICOMP. The focus of my research was to accurately simulate the jet flows and then compute the noise source. Most of these simulations were done using a previously developed axisymmetric jet code. This code uses Gottlieb & Turkel's 2-4 Scheme and was validated[1] by comparing simulation results with the predictions of the linear theory. Large eddy simulations were then done using Smagorinski's subgrid turbulence model. In these simulations[2], wave-like feature of the large scale structure was demonstrated for random inflow disturbances. These structures were then enhanced by imposing harmonic disturbances to the inflow and the sound source was computed. The axisymmetric code was then extended to a round jet (3D) code.

One other issue that I examined was the outflow boundary condition. This is an important element in the numerical formulation. However, reflections at the boundary are very common in numerical simulations. These reflections at the boundary contaminate the numerical solution inside the computational domain. A few boundary conditions, such as Thompson, Bayliss and Turkel, Giles, Hagstrom and Hariharan, Scott and Hankey, etc., were examined. Studies were carried out for unexcited plane jets[3] and excited axisymmetric jets[4]. Issues relating to stretching and filtering near the outflow were also examined.

References

[1] Hayder, M.E., Turkel, E. and Mankbadi, R. R. (1993), "Numerical Simulation of a High Mach Number Jet", AIAA paper 93-0653, Presented at the 31st Aerospace Sciences Meeting and Exhibit, Reno NV.

[2] Mankbadi, R. R., Hayder, M. E. and Povinelli, L. A., (1993), "The Structure of Supersonic Jet Flow and Its Radiated Sound", AIAA paper 93-0549, Presented at the 31st Aerospace Sciences Meeting and Exhibit, Reno NV.

[3] Hayder, M.E. and Turkel, E., (1993), "High Order Accurate Solutions of Viscous Problems", AIAA paper 93-3074, Presented at the 24th Fluid Dynamics Conference, Orlando, FL.

[4] Scott, J.N., Mankbadi, R. R., Hayder, M.E., and Hariharan, S. I., (1993), "Outflow Boundary Conditions for the Computational Analysis of Jet Noise", AIAA paper 93-4366, Presented at the 15th AIAA Aeroacoustics Conference, Long Beach, CA.

DUANE HIXON

The prediction of jet noise has been an active area of research for many years. In the past, Lighthill's acoustic analogy was incorporated into empirical noise prediction codes such as the MGB code developed by Mani, Gliebe, and Balsa. In this work, performed in collaboration with Dr. Reda Mankbadi, a Large Eddy Simulation Navier-Stokes solver is used to directly calculate the nonlinear near field rather than model it. This solution is then matched to an outer solver which uses a modified Kirchoff's method to predict the far-field noise.

The modified Kirchoff's method uses the outer boundary pressure distribution calculated by the LES code as an inner source surface, and the far-field pressure is analytically computed. With this method, the noise at any far-field point may be computed by numerically integrating an analytical expression. This work is still in the beginning stages.

LIN-JUN HOU

The steady state Navier-Stokes equations for the incompressible viscous flow are solved by the Least-Squares finite element method. Comprehensive numerical solutions of the flow over a three-dimensional backward-facing step up to Re=800 are studied. The calculated values of the primary reattachment length are in good agreement with experimental results [1].

Current work is to solve the time-dependent Navier-Stokes equations based on a velocity-pressure-vorticity formulation discretized by backward finite differencing in time. The time-accurate scheme is applied for each time step. Since the system is of first order, only the Dirichlet boundary conditions are needed. Numerical examples of two-dimensional viscous flow over a square obstacle at Reynolds number 200, and flow around a rectangular cylinder at Reynolds number 25 were tested and the results compared well with steady state solutions. Further work is undergoing for higher Reynolds numbers where periodic vortex shedding exist, and the study of the effects of outlet boundary conditions.

Reference

[1] Jiang, Bo-Nan; Hou, Lin-Jun; and Lin, Tsung-Liang: "Least-Squares Finite Element Solutions for Three-Dimensional Backward-Facing Step Flow". Presented in the Fifth International Symposium on Computational Fluid Dynamics, Sendai, Japan, August 31- September 3, 1993.

P. G. HUANG

The present study consists of two parts: implementation of a two-scale model by Liou and Shih [1993] into the compressible code by Huang and Coakley [1992] and testing of the model in flows with complex shock/boundary-layer interactions.

The first part of the work, i.e. the implementation of the two-scale model into the compressible code, has been completed. The model has been tested successfully on a flat plate flow with zero pressure gradient for Mach numbers ranging from 0.1 to 5.

The second part of the work has been partially completed. We have tested the model on Settles' 24 degree compressible corner flow, selected based on a recent review article on supersonic/hypersonic experiments [Settles and Dodson, 1993]. The preliminary results have shown that the two-scale model produces results similar to those of the k-ε model, although skin friction in the recovery region is slightly higher.

The outcome of the study will be submitted to the International Conference on Flow Interaction, to be held in Hong Kong, September 5-9, 1994.

References

[1] Liou, W. W. and T-H Shih: "A Multiple-Scale Model for Compressible Turbulence Flows," ICOMP No. 93-7, NASA TM-106072, 1993.

[2] Huang, P. G. and Coakley, T. J.: "An Implicit Navier-Stokes Code for Turbulent Flow Modeling," AIAA-92-0547, 30TH Aerospace Sciences Meeting, Reno, Nevada, 1992.

[3] Settles, G. S., Gilbert, R. B. & Bogdonoff, S. M.: "Data Compilation for Shock Wave/Turbulent Boundary Layer Interaction Experiments on Two-Dimensional Compression Corner," Princeton University Report 1489-MAE, Princeton University, 1980.

[4] Settles, G. S. and Dodson, L. J.: "Hypersonic Shock/Boundary-Layer Interaction Database," NASA CR 177577, 1993.

BO-NAN JIANG

Work continued on the least-squares finite element method (LSFEM). It is well known that the boundary conditions to be applied to the Navier-Stokes equations have been the subject of constant controversy, and for non-standard boundary conditions there were very few rigorous analyses available. Usually the theoretical analysis of the Navier-Stokes equations is conducted via the Galerkin method (which leads to difficult saddle-point problems) mainly for the standard velocity boundary conditions. We find that the least-squares method based on the first-order differential equations is not only a powerful technique for numerical solution, but also a useful tool for theoretical study of the div-curl equations and the Navier-Stokes equations. We have proved that the div-curl equations which traditionally are considered as an overdetermined system and the Navier-Stokes equations in the first-order velocity-pressure-vorticity formulation are properly determined. The number of boundary conditions on a fixed boundary for the Navier-Stokes equations is four (three in most cases, if the dummy variable $\phi = 0$ on Γ is not counted as the fourth one) for three-dimensional problems, and two for two-dimensional problems. Four different combinations of non-standard boundary conditions are rigorously proved to be permissible for the Navier-Stokes problems. Consequently, the corresponding LSFEM has an optimal rate of convergence for all knowns. The least-squares method and the div-curl method are systematic and consistent methods to obtain a high-order derived version of the differential equations without generating spurious solutions. Specifically, the self-adjoint second-order differential equations obtained by the least-squares method automatically satisfy the divergence-free equations and are suitable for any boundary conditions.

Work has been initiated with S.T.Yu on the simulation of chemically reacting flows by using the least-squares finite element method. A burner-stabilized premixed flame is successfully simulated by the LSFEM that incorporates comprehensive real gas property models. A finite rate chemistry model with eight species and eighteen reaction steps is adopted for the H_2/O_2 reaction. Detailed thermodynamic and transport properties are included for the multiple-species gas mixture.

ARNE V. JOHANSSON

The cooperation between CMOTT and the Royal Institute of Technology has been started off in the area of turbulence modelling and numerical simulation of turbulence and transition. Several common points of interest were penetrated during the two-week visit, although the main focus was devoted to the study of general formulations of constitutive relations, with application to turbulence modelling. The particular area of interest for such general formulations of constitutive relations is that of algebraic Reynolds stress models. Some recent efforts in the direction of constructing explicit algebraic Reynolds stress models have shown promising results. The present work of clarifying some of the fundamentals for such relations has partly been written up during this visit. In addition to completing this effort, continued collaboration is planned in (e.g.) the direct numerical simulation area.

KAI-HSIUNG KAO

It is proposed to develop a detailed method to study the aerodynamic behavior of complex configurations, such as forebody/inlet combinations, propulsion system integration, high speed aircraft, and so forth. The grid construction for the proposed work would initially seem tedious; however, a composite grid system will be employed to greatly simplify the grid generation. Adoption of the CHIMERA scheme allows discretization of the volume about the complex configuration as a collection of structured grids. The overlapped grid approach is used so that individual component grids can be generated easily if one does not have to match to other grids or pieces of geometry. This advantage greatly reduces the tedium of trying to generate a complex 3D grid to define both external and internal flows.

The Chimera overset grid scheme has been successfully applied in combination with a Navier-Stokes solver. The flow solver applies a time accurate, 3D, finite volume, high resolution scheme to calculate compressible full Navier-Stokes equations. Good qualitative numerical results have been obtained in M5 Waverider and Space Shuttle Orbiter simulations. Solution-adaptation enhancement is also performed by using a secondary fine grid system which oversets a base grid in the high-gradient region. Salient features are well resolved for various shock reflection and shock wave/boundary layer interaction problems.

At present, the CHIMERA grid scheme is used to simulate the heat transfer and flow in a simulated turbine blade internal cooling passage configuration. In addition, the grid generated and the methodology for an HSCT fuselage/wing/diverter/nacelle geometry were also examined which will provide a baseline for research into optimization of HSCT aerodynamics, propulsion integration, and sonic boom.

By using the proposed numerical techniques, the preliminary design of complex bodies can be accelerated and made much more efficient. Design improvements will be suggested upon thorough examination of the computational results.

B.P. LEONARD

During 1993, my ICOMP research has continued to be concentrated in three areas: (1) fundamental CFD algorithm development, (2) engineering-accuracy turbulence modeling, and (3) technical politics (relating to numerical uncertainty analysis). Although important advances have been made in items (1) and (2), probably the most important breakthrough is in category (3) — in fact, 1993 will go down as a land-mark year in terms of improving standards of accuracy in CFD publications and conference presentations. The following outline gives somewhat more detail.

(1) CFD algorithm development

The publication of NASA TM 106055 (ICOMP-93-05) summarizes part of my work at ICOMP developing higher-order methods over the past seven years. This report describes the genuinely multidimensional, uniformly third-order polynomial interpolation algorithm (UTOPIA) and related convection-diffusion schemes. In collaboration with Malcolm MacVean and Adrian Lock at the U.K. Meteorological Office in Bracknell, England, I was able to construct the first truly multidimensional flux-limiter giving strictly shape-preserving results. This idea appears to be catching on in the meteorological community, where it's important to have positivity-preserving results. Although the current form of UTOPIA requires Courant number less than one, I have shown that explicit one-dimensional advection schemes don't require this — NASA TM 106203 (ICOMP-93-14). Roughly speaking, since everything is known about a solution at the "current" time-level, the so-called CFL condition (that the numerical domain of dependence should include the physical domain of dependence), although true, is irrelevant — since the numerical domain is the whole field. The ICOMP report gives details for a von Neumann stability analysis. This work has been accepted for publication in *Computer Methods in Applied Mechanics and Engineering.* I have spent a good deal of time on a behind-the-scenes controversy regarding the distinction between finite-difference and finite-volume formulations. Although this, at first, may seem fairly academic, the truncation error is significantly different in the two cases. For example, classical central differencing for diffusion is twice as accurate in an FV formulation as it is in an FD formulation (the exact same operator). There seems to be a general trend in favour of finite-volume methods. The controversy (relating to the order of accuracy of the QUICK scheme) has — hopefully — been laid to rest in a forthcoming ICOMP report. The controversy was initiated by an incorrect publication in *Communications in Applied Numerical Methods* about a year ago. So far, the editors have not been willing to publish my response. However, negotiations are continuing.

(2) Practical turbulence modeling

The workshop on turbulence modeling held here in September demonstrated the strong need for an algebraic turbulence model giving engineering accuracy at reasonable cost. Many "users" emphasized the need for such a model. By default, the zero-equation Baldwin-Lomax model is often used — even though it often gives "funny" results. Turbulence "modelers", on the other hand, tend to look upon zero-equation models as being too unsophisticated to warrant serious attention. However, as my former students, Mark Potapczuk, Nick Georgiadis, and Julie Conley, demonstrated at the workshop, the Modified Mixing Length (MML) algebraic turbulence model, developed at The University of Akron in collaboration with Professor Jerry Drummond, is a viable tool — providing results of engineering accuracy at far less cost than two-equation models that give comparable results. Mark has made several improvements to the original model in his external-flow code for icing research (unstructured grids, surface roughness, etc.). Nick and Julie have made a number of other refinements, using PARC and PROTEUS, respectively. So far (having studied a number of test cases), we have found that MML performs at least as well as Baldwin-Lomax (when the latter is well behaved) and it gives reasonable results under conditions for which Baldwin-Lomax gives bizarre unphysical results or breaks down completely. We continue to encourage "users" to switch from Baldwin-Lomax to MML — the algorithmic structure is virtually the same, requiring (literally) only a handful of lines of recoding. Please contact Mark, Nick, or Julie for coding details.

(3) Breakthrough in accuracy standards

A very significant development for improving the standard of accuracy of CFD computations occurred with the publishing of the Editorial Policy Statement of the ASME *Journal of Fluids Engineering*, September 1993, Volume **115**, pages 339-340. This is something I've been advocating for a number of years, working with Chris Freitas, Ismail Celik, Pat Roache, and others. Chris, Ismail, and the *JFE* editorial board deserve much credit for taking this bold step. As I contributed (rather heavily) to the final version of the adopted editorial policy, I would like to include the "short-list" here:

The Journal of Fluids Engineering will not consider any paper reporting the numerical solution of a fluids engineering problem that fails to address the task of systematic truncation error testing and accuracy estimation. Authors should address the following criteria for assessing numerical uncertainty.

1. The basic features of the method including formal truncation error of individual terms in the governing numerical equations must be described.
2. Methods must be at least second order accurate in space.
3. Inherent or explicit artificial viscosity (or diffusivity) must be assessed and minimized.
4. Grid independence or convergence must be established.
5. When appropriate, iterative convergence must be addressed.
6. In transient calculations, phase error must be assessed and minimized.
7. The accuracy and implementation of boundary and initial conditions must be fully explained.
8. An existing code must be fully cited in easily available references.
9. Benchmark solutions may be used for validation for a specific class of problems.
10. Reliable experimental results may be used to validate a solution.

[The "long-list" can be found on page 340 of the above reference.] One should note, in passing, that this accuracy policy will *exclude* most commercial codes (!). Also, the ASME appears to be enforcing consistent standards in CFD conference presentations.

WILLIAM LIOU

My research involves the development of turbulence models for incompressible/compressible flows and their validation/application in flows of engineering interest. In this reporting period, a compressible two-scale turbulence model was successfully developed and was validated in compressible shear layers, compressible boundary layers, and shock/turbulent boundary layer interaction ramp flows. The implicit Navier-Stokes equations flow solver (developed by P. G. Huang and T. J. Coakley (1992)) and the boundary layer equations solver that were used in the model validation process were compared and found to produce mutually consistent results. The results have shown that the two-scale model can predict successfully both the rate of growth of compressible shear layers and the skin friction coefficient of compressible boundary layers. The development and validation of the model have been summarized in two NASA reports (ICOMP-93-11, CMOTT-93-3, NASA TM106113 by Duncan, Liou and Shih and ICOMP-93-07, CMOTT-93-02, NASA TM 106072 by Liou and Shih) that are being reviewed for journal publication.

I was also involved in developing a new vorticity dynamic based model dissipation rate equation. The model is developed from the mean square vorticity equation at large Reynolds number. The development of the model and its applications are reported in a NASA report (ICOMP 93-20, CMOTT 93-08, NASA TM 106177 by Shih et al.) and a paper has been submitted for journal publication.

Research effort in this period also includes performing turbulent free shear flow computations using an eddy viscosity transport equation model initially developed by T. H. Shih. Preliminary calculations show encouraging results. Further model development and flow calculations are underway.

The behavior of linear stability waves in curved mixing layer was also studied in this period. This type of analysis is a key element in the development of the weakly non-linear wave turbulence model. The model is capable of simulating the dynamical behavior of coherent large-scale turbulent structures which dominate the mixing process in most turbulent shear flows. This study has shown that when the shear layer is unstably curved, streamwise vortices, similar to the Gortler vortices developed in the boundary layer along a concave wall, appear. This finding conforms with published experimental observations. The analysis and some results of this study is detailed in a NASA report (ICOMP-93-28, CMOTT-93-11, NASA TM 106290 by Liou). The manuscript has been accepted for publication in the Physics of Fluids journal (February 1994).

CHAOQUN LIU

Flow transition is one of the fundamental and unsolved problems in modern fluid mechanics. The existing numerical studies are quite limited for the following reasons:

1. Most of them use a temporal approach which is non-physical,
2. The codes blow up at the flow breakdown stage before transition (pre-onset simulation only),
3. They are very expensive in cpu cost (around 100 - 1000 CRAY hours for a 3-D flat plate).

A new technology was developed during my visit at ICOMP in May-August, 1993, which led to a successful numerical simulation of the whole process of flow transition at acceptable CPU cost. A fourth-order finite difference scheme on stretched and staggered grids, a fully implicit time-marching technique, a semi-coarsening multigrid based on the so-called approximate line-box relaxation, and a buffer domain for the outflow boundary conditions were all used for high-order accuracy, good stability, and fast convergence. A new fine-coarse-fine grid mapping technique was developed to keep the code running after the laminar flow breaks down. A number of numerical simulations have been performed which show good agreement with linear stability theory, secondary instability theory and some experiments. This potentially provides a tool for direct numerical simulation of bypass transition. The cost for a typical case with 162 x 34 x 34 grid is around 2 CRAY-YMP CPU hours for 10 T-S periods.

JAMES LOELLBACH

My research is focused primarily on the generation of structured grids in support of flow analyses of turbomachinery components using the HAH3D flow solver, developed by Chunill Hah of NASA Lewis. Applications include compressible transonic flows through axial compressor and turbine stages, and incompressible flows through centrifugal pumps. I am also working (in cooperation with Chunill Hah, Fu-Lin Tsung of ICOMP, and Oh Joon Kwon of Sverdrup Technologies) on developing a hybrid structured/ unstructured flow solver.

Turbomachinery component geometries often consist of closely spaced blades at high pitch angles within a single blade row, as well as closely spaced blade rows in the axial direction. The requirements of establishing periodic boundaries between the blades and axial boundaries between the blade rows can produce high skewness in structured grids. Excessive grid skewness degrades both the accuracy and convergence rates of most flow solvers.

In an attempt to alleviate this problem, we are deviating from the H-grid topology often employed in the axial and blade-to-blade directions. Instead, we are using what is called an I-grid topology, which is essentially an H-grid centered about a blade with relaxed periodicity constraints on the interblade boundaries. Points are allowed to move along the curves defining these boundaries until the blade-to-blade grid lines intersect the boundaries at nearly right angles. Thus, while the curves from the grids of adjacent blades that share a common interblade boundary overlay one another, they are discretized differently. The advantage to this approach is that the overall skewness of the grid can be greatly reduced; the flow solver, however, must include additional logic to interpolate flow properties across the interblade boundary.

I have been developing a set of programs and subroutines to simplify the generation of such grids for a wide range of configurations. The goal is not to develop a single, generalized grid generation package, but instead to create a set of routines which can be quickly combined into simple, easy-to-use programs for specific types of geometries. Eventually, some higher-level drivers may be written to link particular routines together for application to broader classes of configurations. The grid generation techniques that are used consist of both algebraic and elliptic differential equation methods.

The use of unstructured grids offers another approach to alleviating some of the problems caused by the complicated geometries of turbomachines. Unstructured grids, composed of tetrahedra or more general polyhedra, can be created using generalized methods for practically arbitrary configurations. Unstructured grid generation and flow solution techniques, however, are not fully mature, especially for three-dimensional viscous flows. Grid generation issues concerning cell aspect ratios and small cell sizes in boundary layers are being addressed by many different researchers. Flow solution issues related to the accuracy of viscous terms and turbulence modeling are also unresolved at this time. By developing a hybrid structured/ unstructured flow solver, we are attempting to take advantage of the strong points of both approaches. A structured grid will be used near solid walls where fine grids and accurate viscous modeling are required; away from solid walls, the structured grid will mate with an unstructured grid. Research on this project is still in the early stages.

ANASTASIOS S. LYRINTZIS

There has been an increased interest in aeroacoustics mainly due to NASA's efforts to develop a high-speed civil transport plane. The success of this important project depends on the substantial reduction of jet exhaust noise. The important features of jet noise have been studied in the past using acoustic analogy. A new approach (i.e. Kirchhoff's method) is investigated here. This is a calculation of the nonlinear near- and mid-field with the far-field solutions found from a linear Kirchhoff formulation evaluated on a surface S surrounding the nonlinear-field. The surface S is assumed to include all the nonlinear flow effects and noise sources. The full nonlinear equations are solved in the first region (near-field), usually numerically, and a surface integral of the solution over the control surface gives enough information for the analytical calculation in the second region (far-field). This method provides an adequate matching between the aerodynamic nonlinear near-field and the acoustic linear far-field. The method is simple and accurate and seems to overcome some of the difficulties of the acoustic analogy approach. This work is currently in progress, and more details can be found in Reference [1].

Reference

[1] Mankbadi, R. R., and Lyrintzis, A. S., "On the Prediction of Far-Field Jet Noise" submitted to the Symposium on Boundary Layer & Free-Shear Flows, ASME Fluids Engineering Division Summer Meeting, Lake Tahoe, June, 1994.

SHERWIN MASLOWE

Vortex pairing was first observed in transitional mixing layers and subsequently found to occur in turbulent free shear layers, as well, where it is believed to contribute significantly to the downstream growth of the mixing layer. My research this year had as its goal the formulation of a rational theory capable of modelling vortex pairing and related three-dimensional phenomena, such as the generation of pairs of counter-rotating streamwise vortices.

An early and relatively successful theory due to Kelly (1967) explained the initiation of vortex pairing as the result of a subharmonic resonance between the equilibrated most unstable wave of linear theory and a subharmonic of, initially, much smaller amplitude. Monkewitz (1988) published an amended version of the theory in which both waves are of comparable amplitude. His paper exhibited an important shortcoming of the basic mechanism which had already been pointed out by Kelly: for plane spatially growing waves, the case pertinent to experiments, the resonance conditions cannot be satisfied exactly due to dispersive effects and the resultant detuning leads to a perturbation expansion of limited validity.

This difficulty can be overcome by choosing the subharmonic component to be a pair of oblique waves inclined at approximately $\pm 60°$ to the mean flow direction. Different possibilities exist depending on the amplitude of the oblique waves relative to that of the plane wave. When all modes are initially of the same order, the oblique waves exhibit very rapid (exponential of an exponential) amplification, whereas the linear growth of the plane wave is unaffected by the presence of the oblique waves; this is termed parametric resonance. A more general development from which parametric resonance is recovered in one limit results from scaling the oblique waves to be slightly larger than the plane wave [$O(\varepsilon)$ vs. $O(\varepsilon^{4/3})$]. This scaling leads to a pair of coupled, nonlinear integro-differential equations for the amplitude which have been derived for the adverse pressure gradient boundary layer by Goldstein and Lee (1992) and the Stokes layer by Wu (1992). These amplitude equations develop a singularity in a finite time (or distance) and the significance of this "explosive instability" is a subject of current research. Some recent unpublished numerical simulations for a free shear layer suggest that a rapid thickening of the layer is one consequence.

A formulation more consistant with some of the experimental observations is presently being developed by Dr. Lennart Hultgren and myself. The oblique waves will initially have smaller amplitudes than the plane wave so that the latter will saturate, as it does when evolving alone. During this stage of the evolution, the critical layer thickness associated with the oblique waves will be larger (due to their rapid amplification) than that of the plane (2D) wave. Evolution equations appropriate to the next stage of the process will then describe the onset of vortex-pairing according to our analysis (which parallels a similar investigation of the adverse pressure gradient boundary layer described in a paper by Wundrow, Hultgren and Goldstein).

A. F. MESSITER

Further work has been done on long-wave spatial instability of a supersonic shear layer, in the case where the disturbance is large enough that the amplitude of the shear-layer displacement is of the same order as the shear-layer thickness. This differs from another important special case considered by T. F. Balsa, in which disturbances of a smaller amplitude, but still characterized by a nonlinear critical layer, can develop through growth of a linearized disturbance imposed further upstream. As in Balsa's study, the analysis here makes use of matched asymptotic expansions and multiple scales, but now the nonlinearity in the external flow appears earlier, on the same large spatial scale as in the critical layer. Moreover, the description of the critical layer now resembles that of Goldstein and

Hultgren [JFM **197** (1988)], but with an additional equation for the temperature. The temperature and vorticity satisfy certain solvability conditions, and two expressions for the velocity jump across the critical layer lead to a quite complicated evolution equation for the disturbance amplitude. Results for instability of a vortex sheet are recovered in the limit as the ratio of shear-layer displacement to thickness becomes large. A report is in preparation.

During a previous summer visit, I had initiated a study of weakly three-dimensional instability of an incompressible shear layer, for a case which appeared to bridge the gap between the two-dimensional study of Goldstein and Leib [JFM **191** (1988)] and the strongly three-dimensional study of Goldstein and Choi [JFM **207** (1989)]. In the new case, the small spanwise wavenumber has a specific order of magnitude corresponding to the first appearance of three-dimensionality (through vortex stretching and the spanwise pressure gradient). D. Davis has extended the formulation, and has begun the adaptation of an earlier numerical scheme for this problem. Discussions about the study have continued, and S. J. Leib has undertaken the completion of the numerical work.

DAVID MODIANO

My research at ICOMP in 1993 was in two areas, which are summarized below.

1. Parallel Processing for Multiblock Multigrid Calculations.

The goal of this project is to develop a parallel multigrid multiblock flow and heat transfer simulation code that is dependable enough to be used in industry. It was decided that a loosely parallel system, such as a network of workstations, would be more attractive to industry than would a dedicated parallel processing system. I implemented a parallel multiblock multigrid solver for Laplace's equation, on the LACE network at NASA Lewis, a cluster of IBM RS/6000 workstations linked by ethernet. One grid block was assigned to each processor, with message passing to establish boundary conditions at block interfaces. I used a block size that would simulate the computation-to-communication ratio of a multiblock grid for turbine cooling passage flow simulation.

The central conflict in this method of parallelism is that while computational efficiency rises with increased block grid size (because computational costs grow faster than communications costs), the multigrid algorithm uses a sequence of coarse grids to improve convergence. I investigated Yoram and Caughey's vertical mode (AIAA Journal, v. 30, no. 8, Aug. 1992) in which message passing occurs only at the fine grid level, with the block boundary conditions frozen at the coarse levels. The communication costs using ethernet were very high. Also, I found that multigrid convergence suffered greatly in vertical mode, more than offsetting the reduced communication. However, this is a model problem for which multigrid typically performs much better than for real flow problems, so the decreased convergence is more noticeable.

I compared two message passing libraries, the Application Portable Parallel Library (APPL), developed at Lewis, and the Parallel Virtual Machine (PVM), initially developed at the Oak Ridge National Laboratory. APPL is more mature, and showed better performance. PVM, which is almost the *de facto* industry standard, requires further development.

2. Adaptive Mesh Refinement (in collaboration with E. Steinthorsson of ICOMP and P. Colella of UC-Berkeley).

The goal of this project is to apply the Adaptive Mesh Refinement (AMR) method to the solution of the Navier-Stokes equations on mapped grids. Previous work by Colella *et al.* produced an AMR algorithm for Cartesian meshes and a Godunov-based solver for the Euler equations on mapped grids. Work at ICOMP in 1994 will involve the extension of the AMR algorithm to mapped grids and the implementation of a scheme incorporating the viscous effects.

ROY NICOLAIDES

Fully automatic triangulation of complex (usually nonconvex) domains has been investigated in detail. An optimal algorithm for this has been developed. This algorithm produces a Bounded Delaunay Triangulation of the domain. The BDT is a new generalization of the Delaunay triangulation which we introduced in order to allow boundary and other edges to be specified as part of the triangulation even if they are not in the standard triangulation. Any such definition requires that the standard triangulation be modified, and it is particularly important to know that illegal edge crossings are not produced. We have proved this result for our method. A comprehensive program was written implementing the BDT. To demonstrate the algorithm, it has been applied to a number of different geometries. The most difficult of these geometries is a turbine coolant passage containing a number of pin fins and other incursions. The algorithm produced the mesh without error or intervention although it did not try to use finer meshes in regions where boundary layers are expected. This aspect is being addressed currently.

ANDREW T. NORRIS

The work performed this year has been mostly on the numerical development of the compressible hybrid PDF (probability density function) code of Hsu [1]. This has involved extensive rewriting of the PDF part of the code, and performing validation tests. Other work has involved model development of the PDF model.

The hybrid PDF code is a coupling between a compressible finite difference flow solver (Rplus code) and a particle based Monte Carlo solver for the joint PDF of species and enthalpy. The advantages of this coupling lie in the ability of the PDF method to solve turbulent chemical reaction exactly. The first version of this code was released earlier this year [1], and since then an extensive overhaul of the code has been performed. The major changes have been to port the code to a UNIX-based workstation environment, include a portable random number generator, implement a new time-averaging scheme and convection routine, enable the use of lookup tables in the reaction calculation and change the turbulent mixing model to take advantage of the new random number generator. The purpose of these changes has been to make the PDF part of the code faster and take less space, without compromising the accuracy of the solution. The new version of the code has been tested for the case of a piloted diffusion flame of $CO/H_2/N_2$ - air, investigated experimentally by Masri et al [2]. Preliminary results indicate that the code performs correctly for this flow, predicting flame blow-off when the fuel jet velocity is increased above a certain value. However further tests are required to validate the performance.

In the current version of the PDF code, the variables solved for are the composition and the enthalpy. To fully determine a thermodynamic system, another state variable is needed, and at present this value is imported from the Rplus code. In order to make the PDF code self contained, it is suggested that a second state variable be modeled, rather than imported. This will provide a particle dependent value for the calculations, rather than a cell averaged value and so provide a more accurate model for the PDF transport equation. Preliminary work has been performed on this, with density selected as the new state variable. Good results have been obtained for a simple ramp shock, but further model development is needed for the density-dilitation term.

References

[1] A. T. Hsu and Y. L. P. Tsai. LPDF2D: Lewis probability density function code for chemically reactive flows. Technical Report Version 1.0, CFD Branch, Internal Fluid Mechanics Division, NASA Lewis Reasearch Center, 1993.

[2] A. R. Masri, R. W. Dibble, and R. S. Barlow. The structure of turbulent, pilot-stabilised flames of $CH_4/CO/H_2/N_2$ fuel mixtures. 1993. in press.

ALEXANDER ORON

This work, in collaboration with Dr. R. J. Deissler of ICOMP and Dr. J. C. Duh of Sverdrup Technology, was focused on the study of Marangoni convection in an extended fluid layer with two free deformable interfaces, as a continuation of the effort initiated a year ago. The model used in the work can simulate instability in a fluid layer lying on top of another heavier fluid layer. Thus, generally, the values of the Marangoni number are different at the two interfaces. The possibility of different surface tensions at the interfaces is also taken into account.

To find out whether the transition from the quiescent state into motion is monotonic or oscillatory, a linear eigenvalue problem has to be solved for a set of ordinary differential equations. For this purpose, a high-precision code using Chebyshev polynomials, was written and tested for several related problems, and the results previously reported in the literature were reproduced.

We have shown that deformability of the interfaces may give rise to the onset of oscillatory instability which manifests itself in propagation of the disturbances along the fluid layer. It was found that in a certain domain of parameters, the onset of oscillatory instability occurs before monotonic instability sets in. In the considered case of infinitesimally small Biot numbers at both the boundaries, the instability has a longwave character. Different asymptotic regimes of the onset of longwave stationary and oscillatory instabilities are studied analytically. Regimes with no oscillatory behavior are also found. A non-linear stability analysis is planned for the near future.

CHRISTOPHE PIERRE

The basic purpose of my research is the development of computational tools for understanding and predicting the effects of unavoidable blade-to-blade dissimilarities, or mistuning, on the dynamics of nearly cyclic bladed-disk assemblies. This topic is of importance as mistuning has been shown to increase the forced response amplitudes of some blades significantly, and even to lead to single blade failure. Furthermore, the trend toward high performance propulsion turbomachinery designed for finite service life demands an accurate prediction of system performance and dynamics.

This past year, I examined the dynamics of mistuned assemblies with several component modes per blade. In particular, the interaction of two blade modes which have close frequencies (for example, a bending mode and a torsion mode, or two modes of a plate-like blade) and the resulting effect on the sensitivity of the assembly to mistuning has been explored. A paper, [Pierre, C., and Murthy, D. V., "Aeroelastic Dynamics of Mistuned Blade Assemblies with Closely Spaced Modes," AIAA Paper 93-1628, proceedings of the 34th AIAA/ASME Structures, Structural Dynamics, and Materials Conference, San Diego, California, April 18-21, 1993] has been written and presented.

RICHARD PLETCHER

Work continued with Philip Jorgenson on the simulation of internal viscous flows using unstructured grids. An implicit cell-centered finite-volume based scheme has been developed for the compressible form of both the Euler and Navier-Stokes equations. Several test cases were studied to establish the merits of the formulation. Details have been reported in Jorgenson's Ph.D. dissertation (Jorgenson, P. C. E., "An Implicit Numerical Scheme for the Simulation of Internal Viscous Flows on Unstructured Grids," Ph.D. Dissertation, Iowa State University, 1992) and can also be found in a paper to be presented at the 1994 Aerospace Sciences Meeting (Jorgenson, P. C. E. and Pletcher, R. H., "An Implicit Numerical Scheme for the Simulation of Internal Viscous Flows on Unstructured Grids," ICOMP 93-48, NASA TM 106437, AIAA Paper 94-0306). Work is currently under way to extend the unstructured grid capability to three dimensions making use of massively parallel computers (with graduate student Tom Ramin). As an initial step, a two-dimensional unstructured scheme has been implemented on a 128 processor nCube2 computer. Work is now proceeding on a cell-centered implicit scheme in three dimensions.

Further work was done toward the low Mach number conditioning of the discretized compressible time-dependent Navier-Stokes equations. Recent work confirms that the approach is both accurate and efficient for time-accurate calculations. At sufficiently low Mach numbers it was possible to reproduce results characteristic of completely incompressible fluids. The most recent results were reported in a paper presented at the 1993 AIAA Computational Fluid Dynamics Conference (Pletcher, R. H. and Chen, K-H., "On Solving the Compressible Navier-Stokes Equations for Unsteady Flows at Very Low Mach Numbers," ICOMP 93-42, NASA TM 106380, AIAA Paper 93-3368).

Collaboration with Lewis personnel is also in progress on the further development of efficient numerical schemes for combustion applications (with graduate student Rob Cupples). Recent work included the development of a multigrid scheme for use with a strongly implicit numerical procedure for solving the two-dimensional Navier-Stokes equations.

STANLEY G. RUBIN, PREM KHOSLA, AND DAVID BROWN

The RNS (Reduced Navier Stokes) viscous flow codes have been restructured for application to two dimensional/axisymmetric (RNS2D) and three dimensional (RNS3D) supersonic inlet, nozzle and exhaust flows. The current activity on RNS2D is related to computations for a M=2.5 axisymmetric inlet that has been tested at NASA Lewis and for which experimental data and PARC computational solutions are available for comparison, for a nozzle to be tested at NASA Langley, and for the shear layer interaction associated with two exhaust streams, for which PARC and RPLUS results are available. The objective is to minimize CPU for design application, to upgrade PNS methodology for flows with recirculation and strong shock interaction, and to compare RNS accuracy and efficacy against other time dependent Navier Stokes codes. The Lewis inlet conditions and data have been provided by Dave Saunders, the Langley interaction is with John Korte and Ajay Kumar, and the shear layer interaction is with Hien Lai and D. R. Reddy. For RNS3D, the code is being modularized for ease in general application, turbulence models are being introduced, and improvements in iterative efficiency are under investigation. The code is to be tested on a rectangular inlet at various free stream conditions and compared with PARC computations and available data.

During the period of this ICOMP visit, several elements of this research program were initiated while others were at various stages of development or completion. RNS2D solutions were obtained for a number of geometries and flow conditions. These are now summarized: (i) Solutions were obtained for the axisymmetric inlet at Reynolds numbers of 10^6 and $2.5\text{x}\ 10^6$, back pressure ratios of 1.9 and

5.5, and with 0 and 2% bleed in the vicinity of the centerbody expansion corner. Shock-boundary layer interaction due to centerbody, cowl lip and possible terminal shock formation were captured and the overall quality of the results is quite good; (ii) solutions were obtained for high Mach number (7-10) shock-boundary layer interaction due to a compression corner in an unbounded flow and in an inlet. For the later case, the RNS calculations provide solutions with upper wall recirculation; (iii) solutions were obtained for the exhaust shear layer interaction. Agreement with more time consuming PARC and RPLUS computations was quite good. Moreover, only the RNS2D computations provided convergence to machine accuracy. For RNS3D, both Baldwin-Lomax and k-ε turbulence models were introduced into the code. Preliminary results were obtained for backstep channel and curved duct geometries. Application to the rectangular inlet is the next element to be considered.

ROBERT RUBINSTEIN

1. Nonequilibrium and time dependent turbulence modeling

Nonequilibrium turbulence is turbulence which is not close to a Kolmogorov steady state. It is characterized by large imbalance between production and dissipation and is observed in near wall turbulence and in transient homogeneous shear flow at high strain rates.

The goal of work in this area is to derive models of nonequilibrium turbulence starting from the direct interaction approximation. This theory is the only existing fully transient theory of turbulent spectral dynamics. It is therefore a natural foundation for an investigation of this type.

It has been shown that both the Launder-Reece-Rodi model and a modified rapid distortion theory model of Townsend, Hunt, and Maxey are special cases of this theory. They arise from a Markovian specialization; by retaining the non-Markovian character of the direct interaction approximation, closer agreement with rapid distortion theory in regimes of very high strain rate should be obtained.

Current investigations are focused on transient homogeneous shear flow at very high strain rates. This flow has resisted description by existing models.

Publication: "Time dependent turbulence modeling and the direct interaction approximation," ICOMP 93-41, CMOTT 93-15, NASA TM 106379, in review for Physics of Fluids A.

2. Local and distant interactions in turbulence

Discussions with Y. Zhou, G. Vahala, and D. McComb at the Langley summer workshop on turbulence and transition led to reconsideration of the role of the distant interaction approximation in the Yakhot-Orszag theory. The expansion about a logarithmic theory dominated by distant interactions used in that theory can be understood as an expansion in the strength of local interactions.

Publication: "The Yakhot-Orszag theory and local interactions," Proceedings of the 1993 Summer Workshop on Turbulence and Transition, NASA Langley.

3. Boussinesq turbulence

Recent experiments on buoyant turbulence at very high Rayleigh number have suggested the applicability of Bolgiano scaling in such flows. Renormalization group analysis of this regime is an interesting possibility because it can predict amplitudes associated with this theory as well as scaling exponents and lead to models for calculating such flows. At present, a procedure for deriving the scalings has been developed and evaluation of amplitudes (constants in spectrum laws, eddy diffusivity constants) is in progress.

WILLIAM W. SCHULTZ

Downstream computational boundaries remain a source of difficulties, especially in unsteady problems such as the aeroacoustics problems of a round jet. During my visit, we examined the possibility of using sponge-layers as proposed by Israeli and Orszag (1981). Discussions with John Goodrich, Reda Mankbadi and Ehtesham Hayder showed that a simple dissipative layer may have worked better than a boundary condition based on the method of characteristics. It should be possible to use these characteristics to preferentially dampen the incoming waves using these sponge layer approaches.

JAMES N. SCOTT

Research has been conducted on the formulation and implementation of boundary conditions for the numerical simulation of unsteady jet flow in conjunction with computational analysis of jet noise. The primary objective of this effort has been to examine various boundary condition formulations for radiation and outflow boundaries which preserve the wave-like behavior of the unsteady jet structure. Particular attention has been directed toward approaches that ensure that the acoustic disturbances will pass through the boundaries without distortion, amplification, or damping. In addition, only methods are considered which avoid non-physical reflections at the boundaries. As a result of this effort, six (6) candidate forumulations have been identified for implementation into Navier-Stokes codes for unsteady jet flow analysis. Various attributes of the different formulations are compared with one another and with previous approaches. The details of these comparisons and the numerical results of the jet flow computations are described in a forthcoming ICOMP report and an AIAA paper to be presented at the 15th Aeroacoustics Conference, October 25-27, 1993 (i.e. AIAA 93-4366 "Boundary Conditions for Numerical Simulation of Jet Noise")

AAMIR SHABBIR

Modeling of the scalar field of a turbulent flow has lagged the modeling of its velocity field. The most common approach uses a two equation model to calculate the turbulent eddy viscosity for the turbulent velocity field but assumes a constant value of the turbulent Prandtl number to calculate the turbulent scalar diffusivity. Obviously this approach has limitations in situations where the turbulent Prandtl number is not constant.

To overcome this shortcoming, some recent studies have proposed new models in which transport equations for scalar variance and its dissipation rate are solved to calculate the thermal diffusivity. These include, among others, the work of Nagano and Kim[1] and Youssef, Nagano and Tagawa.[2]

The present work is aimed at developing a new two equation model for the scalar turbulence in which transport equations for the scalar variance and its dissipation rate are used to calculate the turbulent scalar flux. The work toward the developing a new model equation for the thermal dissipation equation for wall-free flows has already been completed. A new model for the constitutive relation for the scalar flux has also been developed. The complete model has been successfully applied to several benchmark cases of homogeneous scalar turbulence. Currently efforts are under way to extend the model to the wall bounded flows.

The present work differs from the other work in the following two respects. (1) In other work, the extension of the scalar dissipation rate is based upon the work of Newman *et al.*[3] who developed the production/destruction mechanisms of the thermal dissipation equation in an analogue fashion to those of the mechanical dissipation rate equation. The model equation proposed in the present study is based on the exact transport equation for scalar dissipation and, its production/destruction mechanisms differ from those proposed in the other studies. (2) The model coefficient in the scalar flux constitutive relation used in the present study is not a constant but is a function of the local invariants.

Although the newly developed scalar dissipation equation is used in the present study at the two equation level, its use is not restricted to this level of models. It can be used at any level of modeling which requires an equation for this variable. Therefore, in future studies we will be using this new model equation at the second order level. Another ongoing effort at ICOMP is to propose models for bypass transitional flows. The κ-ε model modifications for the velocity field of such a flow have already been proposed and tested by Yang and Shih.[4] The models proposed here will form the basis for the scalar part of such bypass transitional flows.

References

[1] Nagano, Y. and Kim, C.: "A two equation model for heat transport in wall turbulent shear flows", J. Heat Transfer, 110, 583-589, (1988).

[2] Youssef, M. S., Nagano, Y. and Tagawa, M.: "A two-equation heat transfer model for predicting turbulent thermal fields under arbitrary wall thermal conditions", Int. J. Heat Mass Transfer, 35, 11, 3095-3104, (1992).

[3] Newman, G. R., Launder, B. E. and Lumley, J. L.: "Modeling the behaviour of homogeneous scalar turbulence", J. Fluid Mech., 111, 217-232, (1981).

[4] Yang, Z. and Shih, T. H.: "A κ-ε model for turbulent and transitional boundary layers", Proceedings of the International conference on Near-Wall Turbulent Flows, Edited by R. M. So, C. G. Speziale and B. E. Launder, 165-175, (1993).

SHYUE-HORNG SHIH

Theoretically, the radiated sound of an exhaust jet can be directly obtained by solving the time dependent, compressible Navier-Stokes equations. However, to obtain the noise for the technologically important high Reynolds number flows is beyond the current computational capabilities. The present approach is that of using the unsteady Navier-Stokes equations to obtain the fluctuating sound source in the near field, then relating the far-field noise to the near-field sources through an acoustic analogy, or by patching the resulting near field to linearized Euler equations describing the far field.

Current effort is focused on the accurate prediction of the unsteady flow field. Time dependent results are obtained by solving the full Navier-Stokes equations using the computer code developed by Dr. M. E. Hayder (ICOMP), Professor Eli Turkel (Tel Aviv University/ICOMP) and Dr. Reda R. Mankbadi (NASA Lewis). Computational results were obtained for a Mach 1.5 supersonic jet with axisymmetric inflow excitation and compared very well with those of the linear stability theory. The disturbances grow spatially and the results show the nonlinear development of supersonic instability waves in the shear layer. Results also demonstrate that the amplitudes of disturbances for various inflow excitation levels have the same peak value, and the location of this peak moves upstream when the inflow excitation level is increased. A well-defined peak was obtained in the spectra of the velocity signal, corresponding to the frequency of the inflow excitation.

At present, research is underway in an attempt to refine the treatment of flow near the singular line (r=0). Work is also concentrated on the introduction of 3-D excitation to the shear layer to study its effects, and the implementation of large eddy simulation of round jet.

TSAN-HSING SHIH

As technical leader of the Center for Modeling of Turbulence and Transition (CMOTT), I have been involved in all the main research activities at CMOTT. The research objective of CMOTT is to improve and/or develop turbulence and transition models for propulsion systems. The flows of interest in propulsion systems can be both compressible and incompressible, three dimensional, bounded by complex wall geometries, chemically reacting, and involve "bypass" transition. The most relevant turbulence and transition models for the above flows are one- and two-equation eddy viscosity models, Reynolds stress algebraic- and transport-equation models, pdf models and multiple-scale models. All these models are classified as one-point closure schemes since only one-point (in time and space) turbulent correlations, such as second moments (Reynolds stresses and turbulent heat fluxes) and third moments are involved. In computational fluid dynamics, all turbulent quantities are one-point correlations. Therefore, the study of one-point turbulent closure schemes is the focus of our turbulence research. However, other models such as the renormalization group theory, the direct interaction approximation method and numerical simulations are also pursued to support the development of turbulence modeling.

The details about my own research work are described in the 1993 Research Briefs of CMOTT and in the Proceeding of 1993 Lewis Internal Workshop on Turbulence Modeling.

WEI SHYY

The major activity during the past year was to investigate accurate estimations of numerical fluxes by considering the treatment of convective and pressure fluxes as two separate entities. Employing this idea, Liou and Steffen (Liou 1992, Liou & Steffen 1993) have proposed a scheme, called AUSM (Advection Upstream Splitting Method) which treats the convective terms and the pressure terms separately. In the AUSM scheme, the interface convective velocity is obtained by an appropriate splitting and the convected variable is upwinded based on the sign of the interface convective velocity. The pressure terms are handled using an appropriate splitting formula. Thakur and Shyy (1992, 1993, Shyy & Thakur 1993) have also developed a scheme called the Controlled Variation Scheme (CVS) in which the convective flux is estimated using Harten's second-order TVD scheme (modified flux approach) and the pressure flux is split in a manner similar to the AUSM scheme. For both the CVS and AUSM schemes, the operation count is substantially smaller compared to Roe and Osher schemes since both CVS and AUSM do not entail the evaluation of either Jacobian matrices or intermediate states. Several test cases including shock waves in one and two dimensions and a longitudinal combustion instability problem have demonstrated that the AUSM and CVS schemes yield accuracy comparable to the Roe scheme while being computationally efficient. Further investigation of these schemes for more complex flow fields will be conducted.

References

[1] Liou, M.-S. (1992), "On a New Class of Flux Splittings", *13th Intern. Conf. Numer. Meth. Fluid Dynamics*, Rome.

[2] Liou, M.-S. & Steffen, C. J. (1993), "A New Flux Splitting Scheme," *J. Comp. Phy.*, to appear.

[3] Shyy, W. & Thakur, S. S. (1993), "A Controlled Variation Scheme in a Sequential Solver for Recirculating Flows. Part I. Theory and Formulation", *Numer. Heat Trans.*, to appear.

[4] Thakur, S. S. & Shyy, W. (1992), "Unsteady, One-Dimensional Gas Dynamics Computations Using a TVD Type Sequential Solver," *28th AIAA/SAE/ASME/ASEE Joint Propulsion Conference*, Paper No. 92-3640.

[5] Thakur, S. S. & Shyy, W. (1993), "Development of High Accuracy Convection Schemes for Sequential Solvers," *Numer. Heat Transfer.*, **23B**, 175-199.

ERLENDUR STEINTHORSSON

As in the previous year, the objective of my research has been to develop effective methodologies and computer codes for use in detailed simulations of flow and heat transfer in turbine-blade coolant passages, and other complicated geometries. This year the research has been conducted on two fronts: The continued development of the TRAF3D code, and the use of solution adaptive grid refinement in structured body-fitted grid systems.

Development of TRAF3D: Last year, the first phase in the development of the TRAF3D code was completed. This was the transformation of the code from a highly specialized flow solver for turbine cascade flows to a general single-block flow solver. This year, the second phase in the development of TRAF3D was completed, namely the implementation of a multi-block capability into the code to enable simulations of flows in complex geometries. This latest version of TRAF3D is called TRAF3D.MB. The multi-block capability implemented in TRAF3D.MB is completely general in the sense that there is no limitation on the number of blocks or how they can be connected. For storage, a single one-dimensional array is used for all the individual blocks, with pointers to the first element in each block. The code is configured to automatically order the blocks into memory. If enough memory is available, all blocks are stored in memory at all times. Otherwise, the code automatically separates the blocks into groups or clusters that fit into memory, one at a time. Block clusters that are not in memory are saved in a file (SSD). The code is described in details in ICOMP 93-34, NASA TM 106356.

This year, the third phase in the development of TRAF3D.MB has been started. This is the implementation of an advanced turbulence model into the code. This work is done in cooperation with Dr. Ali Ameri (Resident Research Associate, IFMD). Also, exploratory work on adapting the code for use on parallel computers has been started. This work is done in cooperation with Dr. David Modiano (ICOMP). Both these efforts will continue next year.

Mesh Refinement in Body-Fitted Grid Systems *(it in cooperation with Dr. David Modiano, ICOMP, and Professor Phillip Colella, Univ. of California, Berkeley)*: This year, research into the use of solution adaptive mesh refinement in structured body fitted grid systems was started. The mesh refinement algorithm that best compliments the use of structured, multi-block grid systems is the AMR (Adaptive Mesh Refinement) algorithm of Berger and Colella (*J. of Comp. Physics*, Vol. 87, pp. 171-200, 1990). In this algorithm, the cells on each level of refinement are organized into a small number of topologically rectangular blocks, each of which can be treated as any other member of the multi-block grid system. The small number of blocks leads to relatively small overhead due to management of the grid hierarchy. The size and regular shape of the blocks allow simple and efficient array data structures to be used for the data in the blocks, thus promoting computational efficiency.

The AMR algorithm has been used previously for Cartesian grid systems but will here be extended to curvi-linear body-fitted grid systems. Initially, a single-block flow solver for two-dimensional flows will be developed. Next year, the objective is to extend the methodology to three-dimensions and implement in a multi-block flow solver.

J. T. STUART

In the theory and calculation of the noise radiated from jet flows into the surrounding atmosphere, the concept of the "Green's Function" is very important for relating the pressure fluctuations in the jet to the sound field at much larger distances. A study has been made of the possible choices of Green's Function and of the local treatment of the jet's pressure fluctuations in order to improve the efficiency and optimality of the calculation of radiated noise from a jet.

TIMOTHY W. SWAFFORD

Investigations into impermeable surface boundary conditions associated with the NPHASE code (see below) was the theme of the present effort. This code was originally designed to perform numerical simulations of two-dimensional unsteady cascade flow fields, and while good solutions (in terms of computed surface pressures) can be obtained, excess losses (in terms of entropy generation) have been observed to exist for cases which should be essentially isentropic (e.g., subsonic, inviscid). Therefore, the specific focus of this investigation was to quantify the magnitude and identify the location(s) of these losses for a specific cascade of airfoils, and to make attempts at reducing these losses.

The program NPHASE is a variant of Whitfield's (Mississippi State University) finite-volume implementation of Roe's approximate Riemann solver. The present version of this code can be executed using either the Euler or thin-layer Navier-Stokes equations, although only the Euler equations are considered in the present effort. The numerical scheme is based on formulating the equations as a Newton method where the left-hand-side implicit operator is formed using Steger-Warming flux vector splitting theory whereas the right-hand-side residual is computed using flux difference splitting. Moreover, by interpreting the resulting system of equations as a relaxation method, additional stability is gained by implementing a symmetric Gauss-Seidel iteration procedure within a given Newton cycle. The resulting scheme is observed to be very stable (numerically) and is often executed at CFL numbers exceeding 1000 when considering steady-state solutions.

The geometry chosen was the so-called Tenth Standard Configuration, which consists of a particular NACA airfoil section modified to have a circular arc camber line. The Mach number was 0.7 and the stagger and incidence angles were 45 and 10 degrees, respectively. Only steady-state computations were performed. Surface boundary conditions used in the present version of NPHASE were derived using one-dimensional characteristic variables. The specific objective of this study was to make appropriate modifications of these existing boundary conditions which would reduce the observed entropy generation near the wall. The grid used consisted of 121x41 points in the chordwise and blade-to-blade directions, respectively. Inflow and outflow conditions were obtained using characteristic variables and were not a subject of this study. Periodic conditions upstream and downstream of the airfoil surfaces were enforced using two cells of overlap in the blade-to-blade direction.

As stated above, the initial phase of this effort was spent in identifying and quantifying the origin of entropy generation, which is well known to occur near solid surfaces in most Euler codes.

Moreover, the computation of flow field dependent variables which results in non-isentropic solutions appears to originate at the leading edge. Results for this particular study confirmed this and seemed to be largely confined to the first 2 cells off the wall. This was determined by using the ratio

$$I_R \equiv \left(\frac{p^n}{p_{init}}\right) / \left(\frac{\rho^n}{\rho_{init}}\right)^{\gamma}$$

The end states of density and pressure are those associated with a particular mesh cell at a given time step and are devoted with superscript n, whereas the initial states are those used as initial conditions. Of course, this ratio should be unity for an isentropic process.

As with many Euler codes using a finite volume formulation, boundary conditions are applied using so-called "phantom" (or ghost) cells, where dependent variables in these cells are used as an artifice to enforce whatever conditions are physically called for; in this case, an impermeable surface (no flux through the wall). In an attempt to gain quantitative understanding of how dependent variables within phantom cells associated with a solid surface behave as the solution progresses, the above isentropic ratio was computed in similar fashion as the interior cells discussed above. It was found that entropy generation in the phantom cells (particularly near the leading edge) was of equal magnitude as that observed to occur in the interior cells. As a consequence of this finding, it was decided to focus attention on these cells and attempt to modify the existing characteristic variable boundary conditions (CVBC) such that phantom cell dependent variable updates would take place isentropically, and in so doing, reduce the level of entropy generation within the interior of the computational domain.

Relations for pressure and density which result from impermeable surface CVBC can be written as

$$p_{surface} = p_b = p_r + \rho_o c_o \overline{q}$$

$$\rho_{surface} = \rho_b = \rho_r + \frac{\rho_o}{c_o} \overline{q}$$

$$\overline{q} = \frac{\overline{q} \cdot \overline{A}}{(\overline{A} \cdot \overline{A})^{1/2}}$$

where subscript r refers to "reference" conditions and $\overline{q}$ and $\overline{A}$ are velocity and area vectors, respectively. Also, subscript 0 denotes conditions about which a linearization process was performed in order to derive the CVBC. In current versions of the wall CVBC, reference conditions are taken to be those associated with the first interior cells. In the present approach, a reference "isentropic constant" is computed,

$$C_I \equiv \frac{p_{init}}{\rho_{init}^{\gamma}}$$

For boundary conditions ensuing from the CVBC, we further stipulate that

$$p_r + \rho_o c_o \bar{q} = C_I \left(\rho_r + \frac{\rho_o}{c_o} \bar{q} \right)^{\gamma}$$

where "0" conditiions are computed as before. The above conditions give a nonlinear relation between reference pressure and density, and by using

$$p_r = C_I \rho_r^{\gamma}$$

the resulting equation can be solved iteratively for these reference conditions. In the present effort, a Newton scheme was used to solve for these reference quantities for each phantom cell along both upper and lower surfaces (convergence is achieved in 2-3 iterations).

The result of applying this approach was to essentially eliminate the entropy generation within the phantom cells associated with the airfoil surfaces. However, it was also found that elimination of this source of entropy did not reduce that associated with the interior cells (in fact, it actually increased slightly). It is surprising (and disappointing) that the manner in which phantom cell updates occur has the observed effect on the interior field solution, particularly that near the surface. The reason for this is not clear and warrants further study.

GRETAR TRYGGVASON

Work continued on the development and application of the Tracked, Immersed Boundary (TIB) technique to simulate multifluid flows. The basic method is described in Unverdi and Tryggvason: A Front Tracking Method for Incompressible Flows" (J. Comput. Phys. 100 (1992), p. 25-37). The specific work done at ICOMP focused on drop collisions. When the drops collide, they bounce or coalesce depending on whether the thin film of ambient fluid left between the drops is ruptured or not. The rupturing of such a film in numerical simulations has two aspects. One is the purely technical aspect of changing the topology of the drop boundary (which is explicitly tracked in the TIB method); the second is the modeling of the physics that determines the rupture time. This year, a topology change algorithm was developed for the fully 3D TIB code and the 3D front regridding procedure was improved to make it simpler and more robust. Several simulations showed that the procedure yields "realistic" evolution, including a drainage time for the film and then rapid relaxation to a new drop configuration following the rupture. In these simulations and *ad hoc* criterion was used to determine when to rupture, but the results showed that sometimes it is important to predict the rupture time accurately. A physically based rupture model is therefore necessary. Such model is currently being developed here at Lewis by D. Jacqmin and M. Foster, and we plan to incorporate it into the code.

A manuscript discussing axisymmetric collision, "Head-on Collision of Drops -A Numerical Investigation" (ICOMP Report 93-45, NASA TM 106394 Nov., 1993) has been issued, and a discussion of coalescence of unequal size drops and fully 3D collisions is in preparation.

FU-LIN TSUNG

The current research aims to combine an unstructured solver with a structured solver for turbomachinery applications. The purpose of the work is to take advantage of the efficient structured technology in domains where known high viscosity exist and structured grids can be easily generated, and to incorporate unstructured solvers in regions where its flexibility can be helpful. The project allows direct evaluation of both types of solvers, along with the hybrid procedure, for their pros and cons in turbomachinery applications.

The coupling procedure is currently being tested. The structured solver has been coded to handle both internal and external flows. It is formulated in inertial frame of reference to capture rotational effects, sans the necessity to add centrifugal and Coriolus terms explicitly. (The core of unstructured solver is not written by the present author.)

ELI TURKEL

Work continued on developing and comparing boundary conditions to be used at the inflow and outflow boundaries for jet acoustics problems. A series of radiation boundary conditions developed by Bayliss/Turkel and Giles were compared for both subsonic and supersonic jets. The results appeared in an ICOMP report and at the AIAA fluid dynamics conference in Orlando. (ICOMP Report 93-26, NASA TM 106267, AIAA Paper 93-3024, July 1993). The same report also discusses extensions to the 2-4 Gottlieb-Turkel scheme to make it fourth order accurate in space for the viscous terms in addition to the inviscid terms. The 2D axi-symmetric code has been extended to a full 3D code and work is continuing on the treatment of the singular line, $r=0$. Various treatments are being analyzed and implemented. Another project is the development of preconditioning matrices for incompressible flow. Such a matrix accelerates the convergence to a steady state and can also improve the accuracy of the steady-state numerical solution. One such preconditioner has been implemented in the incompressible cascade code of Andrea Arnone. This work appeared in the AIAA CFD conference in Orlando.

YASUHIRO WADA

During my stay at ICOMP, I studied various flux splitting schemes for compressible flows in collaboration with Dr. Meng-Sing Liou. Through this study, we have developed a new flux splitting scheme which has favorable properties: high-resolution for contact discontinuities; conservation of enthalpy for steady flows; numerical efficiency; applicability to chemically reacting flows. In fact, for single contact discontinuity, even if it is moving, our scheme gives the numerical flux of the exact solution of the Riemann problem. Various numerical experiments including a thermo-chemical nonequilibrium flow indicate the oscillation-free robustness of the scheme for shock/expansion waves.

Reference

[1] Wada, Y. and Liou, M.-S., "A Flux Splitting Scheme with High-Resolution and Robustness for Discontinuities," ICOMP 93-50, NASA TM 106452, AIAA Paper 94-0083, 1994.

DAVID WHITFIELD

During this stay at ICOMP, two separate topics were investigated. The first had to do with trying to decrease the CPU time required to converge solutions (particularly for the linearized Euler equations), and the second had to do with accelerating the convergence of flow equations that contain body force terms. A brief description of each topic follows.

In implicit solution methods, an iterative method is frequently used for the global solution process, but the linear subsystems are usually solved using a direct method. Because these linear subsystems are small (ranging from 3x3 for the two-dimensional incompressible Euler equations to 5x5 for the three-dimensional Euler and Navier-Stokes equations), it is of interest to investigate various ways of decreasing the CPU time required to solve the equations. A method investigated in the past for the two-dimensional incompressible Euler equations [1] is to use a formal Newton formulation where the linear equations are solved using symmetric Gauss-Seidel [2]. The solution of the linear subsystems in the Gauss-Seidel process is solved using a direct method; in this case, Doolittle's compact scheme. However, instead of using Doolittle's scheme directly on the subsystems, what was investigated was to formally take the inverse of the 3x3 diagonal block matrices thus greatly simplifying the solution process. This proved to be an extremely valuable approach when the Jacobian matrix was frozen for some number of cycles and/or several Gauss-Seidel passes were used for the solution of the linear subsystems. Consequently, for the solution to the linearized equations, this proves to be an extremely useful approach because, in this case, the Jacobian matrix is never updated and the inverse of the small diagonal matrices can be taken once and for all, resulting in savings in CPU time for up to a factor of over two. This same approach was applied to the three-dimensional compressible Euler equations. However, the results have not been as good. There is a speed-up, but it is more like a factor of 15% as opposed to 100%. The method presently used to invert the 5x5 diagonal blocks for the three-dimensional problem is not as efficient as was used for the two-dimensional problem and improvements in this area will help improve the time, but it isn't expected to improve it as much as it did for the two-dimensional incompressible flow case.

The second topic investigated had to do with the numerical solution of equations (e.g., the Euler equations) with source terms. One application of the usefulness of this effort would be in some turbomachinery computations where source terms are used to model various aspects of an otherwise complicated physical process. This was investigated using a simple problem consisting of the one-dimensional unsteady Euler equations with varying cross-sectional area. Solutions were obtained with and without the source terms included in the Jacobian. When the source terms were included in the Jacobian, convergence was typically obtained in one-fifth the number of cycles compared to when the source terms were omitted from the Jacobian. Probably more importantly, for extreme area variations, the computations diverged when the source terms were omitted (all the computations were carried for a CFL number of 1000). In some cases, the source terms used in turbomachinery applications are such that the Jacobian cannot be obtained analytically. In this regard, the discretized Jacobian approach used in [2] is thought to be a viable approach.

References

[1] Whitfield, D.L., "Numerical Solution of the Two-Dimensional Time-Dependent Incompressible Euler Equations," Mississippi State University Report in Preparation, September, 1993.

[2] Whitfield, D.L., and Taylor, L.K., "Discretized Newton-Relaxation Solution of High-Resolution Flux-Difference Split Schemes," AIAA Paper No. 91-1539, June, 1991.

DANIEL WINTERSCHEIDT

My research has involved the application of the finite element method (FEM) to both incompressible and compressible fluid dynamics problems. Although somewhat more computationally intensive than traditional CFD methods (finite difference and finite volume), the FEM offers several advantages. The method is inherently unstructured and is therefore able to handle problems involving complicated geometry without difficulty. The FEM also has a sound mathematical basis permitting boundary conditions to be implemented in a very natural way. Because of the success of the FEM in structural analysis, the method is well-suited for combined fluid-structure interaction studies.

For steady, incompressible fluid flow and convective heat transfer, I favor the use of the p-version of the FEM. The p-version of the method achieves the desired level of accuracy by increasing the order of the element approximation whereas the h-version of the method improves the accuracy by mesh (element size) refinement. A combination of mesh and polynomial refinement is refered to as an h-p method. The p-version of the FEM is not as flexible as the h-p version, but it is much simplier to implement. The nature of the p-version element approximation permits the use of different 'p-level' (orders of approximation) in each element. Inter-element continuity of the solution is easily satisfied with the p-version, whereas the h-version requires the use of constraint equations to avoid a non-conforming solution (hanging nodes) at inter-element boundaries.

A new p-version finite element formulation for steady, incompressible fluid flow and convective heat transfer problems has been developed. The weighted residual formulation was obtained by applying the least-squares approach to the transient problem, with the time derivatives approximated by finite differences. In the steady-state limit, this approach leads to a Petrov-Galerkin statement with the time step serving as an upwinding parameter. Numerical results for the 1-D Burgers' equation and 2-D convection-diffusion revealed that when higher order approximations ($p \geq 4$) were used, the most accurate results were obtained with the upwinding parameter set to zero. The method was successfully extended to 2-D incompressible fluid flow, producing accurate solutions with fewer degrees of freedom than conventional h-version methods or the p-version with a 'standard' least-squares formulation.

The approach I have taken for transient, compressible flow has been quite different. As is well known, the compressible equations admit discontinuous solutions in the limit of vanishing viscosity. Thus a general method for compressible flow problems must be capable of efficiently resolving discontinuities that appear in the solution of the compressible Euler equations. These problems pose a severe challenge to the finite element method, which is based on the assumption of solution continuity. It should be noted that extremely sharp solution gradients (or even singularities) can be found in other finite element applications such as stress analysis. However, in these cases the location of the sharp gradients are generally known and can be resolved with h-p refinement. Transient compressible flow involves moving discontinuities and therefore poses a more difficult problem.

Conventional CFD methods (finite difference and finite volume) have developed upwind schemes capable of handling the discontinuous nature of the Euler equations. The schemes include flux vector splitting where the flux terms are split and discretized directionally according to the sign of the associated propagation speeds and flux difference splitting where an approximate local solution of the Euler equations is introduced in the discretization. Most finite element researchers have been reluctant to use these types of upwinding methods, preferring instead to use either Petrov-Galerkin methods or Galerkin methods with some type of artificial dissipation added. Only recently have the above mentioned upwind schemes been incorporated into finite element formulations for the Euler equations.

A common finite element approach for these type of problems involves an explicit Taylor-Galerkin formulation which is basically a finite element implementation of the Lax-Wendroff method.

Linear elements (often trianglular/tetrahedral) are generally used because of the discontinuous nature of the solution and for the simplicity of element computations. Although the method does yield some high frequency dissipation, additional dissipation terms must be added to control strong shocks. The basic scheme is second order accurate although higher temporal accuracy can be achieved using a multi-step method.

My current research involves the development of a finite element method that uses upwinding rather than artificial dissipation. The 'upwinded' flux values are only used in elements with extremely large solution gradients. In smooth regions, the standard Taylor-Galerkin finite element method is used to compute the flux values. The method is therefore second order accurate except in the vicinity of discontinuities. The approach has been implemented for the 1-D Euler equations and successfully applied to shock tube problems. Extension of the method to the 2-D Euler equations is in progress.

ZHIGANG YANG

In the past year, I worked on a number of projects in the area of eddy viscosity models for turbulent and transitional flows. In the following, each of the projects will be briefly described.

1. A Galilean and tensorial invariant $\kappa - \varepsilon$ model for near wall turbulence (with T.H. Shih of ICOMP)

An improvement was made in the $\kappa - \varepsilon$ model for near wall turbulence proposed earlier by us (Yang, Z. and Shih, T. H. "A new time-scale based $\kappa - \varepsilon$ model for near wall turbulence," *AIAA J.*, Vol. 31, 1993.) In the earlier paper, the parameter $Ry = (\kappa^{1/2}y)/v$ was used in the damping function $f\mu$. This parameter is better than the commonly used y^+ in that the model can be used for flows with separation and reattachment. However, R_y the dependence in the damping function makes the model coordinate dependent. It also creates some ambiguity when the model is used for complex geometries, for example, a corner flow. In the present study, this deficiency was overcome by introducing a new parameter R in the damping function. The parameter is defined as $R=\kappa/(S\upsilon)$ where S is the modulus of the strain rate tensor of the mean velocity field. Since this new parameter is expressed in local variables, the model is now Galilean and tensorial invariant and suitable to be used in general CFD code with unstructured grid. A number of benchmark flows were tested to validate the model. The details of the model development and the computational results can be found in Yang, Z. and Shih, T.H., "A Galilean and tensorial invariant $\kappa - \varepsilon$ model for near wall turbulence," ICOMP Report 93-24, AIAA Paper 93-3105 (1993).

2. On the wall functions for two-equation turbulence models (with T.H. Shih of ICOMP)

Near the wall, the turbulence quantities change rapidly. Very fine grids are needed to resolve such a rapid variation. This leads to a substantial increase in the overall number of grid points. In addition the fine grid spacing near the wall means that the grids will be severely stretched, causing some numerical stiffness problem. The use of wall functions, which provide the boundary conditions at the log layer rather than at the wall, obviates all the difficulties mentioned above. However, the existing wall functions were proposed based on the behavior of the turbulent boundary layer over a flat plate at zero pressure gradient and it is known that they are not universally valid.

In the present study, the wall functions commonly used in conjunction with the high Reynolds number two-equation turbulence models are examined for turbulent boundary layers with pressure gradients and are shown to give an inadequate response to the pressure gradient. A new set of wall functions are derived based on asymptotic solutions of the governing equations as the log layer is approached. The new wall functions take into account the effect of the pressure gradient, and reduce to the existing wall functions when the pressure gradient is zero. The conditions under which these new wall functions are derived are quite general. Computations using the new wall functions capture properly the effect of the pressure gradient. This work was reported at the 46th APS/DFD Meeting, November 21-23, 1993, Albuquerque, New Mexico.

3. A vorticity dynamics based model for the dissipation rate equation (with T.H. Shih, W. W. Liou, A. Shabbir, J. Zhu of ICOMP)

A new dissipation rate equation is proposed based on the dynamic equation for the vorticity fluctuation. The production of the dissipation rate in the new equation is always positive. The model is thus expected to be more robust for complex flow calculations when used in conjunction with Reynolds stress models. When used in conjunction with the κ equation, the overall performance is found to be better than that of the standard $\kappa - \varepsilon$ equation. The details of the model development and performance can be found in Shih, T. H., Liou, W. W., Shabbir, A., Yang, Z., and Zhu, J., "Dynamics of vorticity fluctuations and the dissipation rate equation," ICOMP 93-20, CMOTT 93-08, NASA TM 105993 (1993).

4. Flow inhomogeneity and the modeled dissipation rate equation (with T.H. Shih of ICOMP)

The dissipation rate equation is the weakest link in both the $\kappa - \varepsilon$ and the second order closure models. The existing ε model equations are homogeneous models in the sense that except for the transport term, the model form remains the same for both homogeneous flow and inhomogeneous flow. Since the dissipation rate represents the energy flux from the large eddies to the small eddies in the eddy cascade, and the energy containing large eddies are sensitive to the inhomogeneity, it can be expected that the flow inhomogeneity should enter the modeled dissipation rate equation. The exact equation for the dissipation rate also shows that the flow inhomogeneity can increase/decrease the dissipation rate.

In the present study, the effect of flow inhomogeneity in the modeled dissipation rate equation is modeled via the invariant theory approach. ∇S is used to characterize the inhomogeneity of the mean field and $\nabla\kappa$ is used to characterize the inhomogeneity of the turbulent field. The inhomogeneity contribution is assumed to be an extra term in the production/destruction in the dissipation rate equation. While different $\kappa - \varepsilon$ models with the homogeneous dissipation rate model do not respond adequately to the pressure gradient, the present model captures accurately the effect of pressure gradients on boundary layer development. A paper detailing the model development and validation is in process.

5. Modeling of bypass transition (with T.H. Shih of ICOMP)

We have proposed two eddy viscosity models for the calculation of transitional boundary layers which take the effect of intermittency into consideration. Both models are based on a $\kappa - \varepsilon$ model for near wall turbulence proposed earlier by the authors. In the first (and simpler) model, the effect of the intermittency is introduced in the eddy viscosity. In the second (and more complex) model, the effect of the intermittency is introduced in all the terms which are generated due to the turbulence. The modifications to the existing $\kappa - \varepsilon$ model appear in the introduction of the weighting factor in the eddy viscosity (in the first model) or in all the turbulent terms (in the second model). In order to close the above equations, an expression for the weighting factor is needed. We assume the weighting factor is related to both the freestream turbulent level and the intermittency factor of the boundary layer. The intermittency factor is assumed to be determined by the local state of the boundary layer. Further, the shape factor is used to characterize the local state of the boundary layer since both the intermittency factor and the shape factor change monotonically from the laminar boundary layer to the turbulent boundary layer. The simpler model was reported in the International Conference on Near Wall Turbulent

Flows, March 15-17, 1993, Tempe, Arizona. Currently, a paper describing this work is under preparation.

6. A transport equation for eddy viscosity (with P.A. Durbin of CTR and N.N. Mansour of NASA Ames)

We have proposed an eddy viscosity transport model for wall bounded turbulent flows. The proposed model reduces to a quasi-homogeneous form far from surfaces. Near to a surface, the nonhomogeneous effect of the wall is modeled by an elliptic relaxation model. All the model terms are expressed in local variables and are coordinate independent; the model is intended to be used in complex flows. Both parabolic flows and elliptic flows are computed by the present model and the model predictions compare very favorably with the experimental data. The details of model development and numerical results can be found in Durbin, P. A., Mansour, N. N., and Yang, Z., "Eddy viscosity transport model for turbulent flow," which is going to appear in *Physics Fluids A , 1994.*

SHAYE YUNGSTER

Two major CFD research efforts were carried out at ICOMP during the current year. These are summarized below:

1. CFD Evaluation of External Burning.

The computational study of external burning for NASP type nozzles, conducted in collaboration with Charles J. Trefny of NASA Lewis, was completed. This study has produced a valuable computational tool to support experiments conducted at NASA Lewis Research Center. This research is aimed at developing methods to improve low-and high-speed nozzle performance, and to demonstrate transonic drag reduction benefits of external burning for NASP type vehicles.

Two- and three-dimensional CFD codes were developed and applied to study the effects of external combustion on NASP-like nozzles at transonic speeds. A simplified computational model for the external fuel injection and combustion processes was developed, which permitted the use of a two-dimenionsal formulation. This simplification permitted to pursue parametric and scaling studies that would have been otherwise impossible to do using a full three-dimensional approach. (Nozzle flows without external burning, however, still require a three-dimensional approach).

Results obtained for the baseline and extended cowls, with and without a flame holder, were in very good qualitative and quantitative agreement with external burning experiments. A grid refinement study, including the use of adaptive grids, was also completed.

Scaling studies were performed for different flame holder configurations. Parametric studies were conducted at various Mach numbers, in which the effects of nozzle pressure ratio and external burning fuel injection pressure were investigated. Normal and axial thrust forces were calculated and, when possible, compared with experimental values.

This study has improved our understanding of the effects of external combustion on nozzle performance, including issues such as boundary layer separation, shock/boundary layer interactions and the effects of far-field boundary conditions.

The results of this work will be presented at the 32nd Aerospace Sciences Meeting (January 10-13, 1994) in a paper entitled "Computational Study of Single-Expansion Ramp Nozzles with External Burning," (AIAA paper 94-0024).

2. Computation of Shock-Induced Combustion in Methane-Air Mixtures

The purpose of this work, conducted in cooperation with Martin J. Rabinowitz of NASA Lewis, was to develop the computational tools needed to predict accurately the shock-induced combustion of premixed methane-air using a detailed reaction model with as few reactions and species as possible.

This type of flow is found in both airbreathing hypersonic vehicles and ram accelerators. The motivation for this work came from the inability of previous CFD computations to accurately predict these flowfields.

To accomplish this aim, a new technique, Detailed Reduction, was employed in developing a chemical mechanism that was both accurate over a wide range of conditions and practicable in terms of computational resources. The resulting combustion model included 19 species and 52 reactions, and required some modifications to the CFD code structure to fit within the available vectorization limits of the CRAY YMP.

The results of this study were the first accurate predictions of the shock-induced combustion of methane-air mixtures. It was demonstrated that previous calculations were unable to accurately predict these flowfields due to their use of severely limited reduced chemical mechanisms. Therefore, the extra cost involved in using the present mechanism is fully justified.

It was also shown that a broad range of flow and geometric conditions could be calculated without having to "tune" the chemical mechanism each time. The computational resources needed to tune a mechanism are often greater than those required to run an existing model.

The practical use of the present combustion model was demonstrated for a ram accelerator, a device for accelerating projectiles to very high speeds utilizing oblique detonation waves. A novel double-ramp configuration was investigated. This configuration has the advantage of providing more control over the establishment of the detonation or shock-induced combustion process. In addition it was shown that a detonation wave could be stablized without directly interacting with the projectile surface. This could have significant benefits in reducing heat transfer and boundary layer separation.

The results of this study were presented at the 29th Joint Propulsion Conference (ICOMP 93-32, NASA TM 106354, AIAA-93-1917, June 28-30, 1993, Monterey, CA)[1] and at the First International Workshop on Ram Accelerator (Sept. 7-10, 1993, Saint-Louis, France). A paper entitled "Computation of Shock-Induced Combustion Using a Detailed Methane-Air Mechanism" has been accepted for publication in the Journal of Propulsion and Power.

[1]This paper entitled "Numerical Study of Shock-Induced Combustion in Methane-Air Mixtures," by S. Yungster and M. J. Rabinowitz received the 1993 Internal Fluid Mechanics Division best paper award.

JIANG ZHU

A new Reynolds stress algebraic equation model has been developed recently at CMOTT. In this model, the Reynolds stresses are explicitly related to quadratic terms of mean velocity gradients, and the model coefficients are derived through the realizability analysis. An extensive model validation has been made for the following test cases which are of great interest in propulsion systems.

1. Diffuser flows. Two conical diffuser flows are calculated, one with an 8° total angle and the other 10°. In both cases, the flows undergo strong adverse pressure gradients but remain attached. Although the flow configuration looks simple, it is not easy to calculate this type of flow accurately, especially for those boundary layer quantities. The present model is compared with the standard $\kappa - \varepsilon$ model, the RNG-based $\kappa - \varepsilon$ model and the $\kappa - \omega$ model. Comparison shows that these models can generally be listed in the following order of increasing accuracy: the standard $\kappa - \varepsilon$, RNG $\kappa - \varepsilon$, $\kappa - \omega$ and the present model.

2. U-duct flow. This case involves the flow in a 180° planar turnaround duct. It features flow with large streamline curvature. The present model predicts a small separation region at the inner duct wall near the bend exit, which is in line with experimental observations. However, the standard $\kappa - \varepsilon$

model does not predict the flow separation. For the mean flow quantities such as wall friction and static pressure coefficients and velocity profiles, both models give quite similar results.

3. Backward-facing step flows. Two backward facing step flows are calculated, one with a smaller and the other a larger step expansion. Comparisons are made for reattachment points, wall friction and static pressure coefficients and various mean velocity and turbulent stress profiles at different locations. For all the quantities compared, the present model is found to perform better than the standard $\kappa - \varepsilon$ model.

4. Confined jets. Calculations are carried out for confined jets in two ducts, one cylindrical and the other conical with a 5° divergence. In each case, a set of inflow conditions are considered which correspond to flows with no, moderate or strong recirculation. Confined jet flows have features similar to those found in backward-facing step flows such as separation with an unfixed reattachment point and severe adverse pressure gradient, and add additional complexities arising from the motion of the separation point. Detailed comparisons with measurements are made for separation and reattachment points, wall static pressure coefficients, excess flow rate, effective jet width, recirculating flow rate and various mean velocity and turbulent stress profiles. The comparisons show that the present model gives much better predictions than does the standard $\kappa - \varepsilon$ model for almost all the quantities compared.

5. Confined Swirling Coaxial Jets. This is the case of Roback and Johnson. At the inlet, an inner jet and an annular jet are ejected into an enlarged duct. Besides an annular recirculation bubble due to sudden expansion of the duct, a centerline recirculation bubble is created by flow swirling. Both the present and the standard $\kappa - \varepsilon$ models predict the strength of central recirculation and the front stagnation point quite well, but the present model predicts the rear stagnation point much better than does the standard $\kappa - \varepsilon$ model. For the mean and turbulent velocity profiles at different locations, both models are found to give quite similar results.

REPORTS AND ABSTRACTS

Feiler, Charles E., (Compiler AND Editor): "Institute for Computational Mechanics in Propulsion (ICOMP), Seventh Annual Report-1992", ICOMP Report No. 93-1, NASA TM-106155, May 1993, 54 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in Cleveland, Ohio to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1992.

Mawid, M. A. (ICOMP); Bulzan, D. L. (NASA Lewis); and Aggarwal, S. K. (ICOMP): "Structure of Confined Laminar Spray Diffusion Flames/Numerical Investigation", ICOMP Report No. 93-2, NASA TM-106038, February 1993, 28 pages.

The structure of confined laminar spray diffusion flames is investigated numerically by solving the gas-phase conservation equations for mass species, continuity, momentum, and energy and the liquid-phase equations for droplet position, velocity, size, and temperature. A one-step global reaction scheme along with six equilibrium reactions are employed to model the flame chemistry. Monodisperse as well as polydisperse sprays are considered. The numerical results demonstrate that liquid spray flames substantially differ from gaseous flames in their structure, i.e., temperature, concentration, and velocity fields, shape, and dimensions under the same conditions. Spray flames are predicted to be taller and narrower than their counterpart gaseous ones and their shapes are almost cylindrical, in agreement with experimental observations. The numerical computations also show that the use of the equilibrium reactions with the one-step reaction scheme decreases the flame temperature compared to the one-step reaction scheme without the equilibrium reactions and more importantly increases the surface area of the flame zone due to a phenomenon termed "equilibrium broadening". The spray flames also possess a finite thickness with minimal overlap of the fuel and oxygen species. A case, for which a fuel-mixture consisting of 20 to 80 percent gas-liquid by mass is introduced into the combustor, is also investigated and compared with predictions using only gaseous or liquid fuel.

Mawid, M. A. (ICOMP); Bulzan, D. L. (NASA Lewis); and Aggarwal, S. K. (ICOMP): "On the Structure of Gaseous Confined Laminar Diffusion Flames/Numerical Investigation", ICOMP Report No. 93-3, NASA TM-106039, February 1993, 24 pages.

The structure and characteristics of gaseous confined laminar diffusion flames are investigated by numerically solving the time-dependent 2D axisymmetric conservation equations. The numerical model accounts for the important chemical and physical processes involved, including axial diffusion, viscous effects, radial convection, and finite-rate chemistry. The numerical results clearly show that the flame has a finite thickness and leakage of fuel vapor into the flame zone is possible. The effect of heat release is found to induce some radial flow. Predicted flame shape and dimensions are compared to the classical Burke-Schumann flame. The numerically calculated flame is observed to be about 15 percent taller and 5 percent narrower than that of the Burke-Schumann solution under the same conditions.

Hagstrom, Thomas (ICOMP); Hariharan, S. I. (ICOMP); and MacCamy, R. C. (Carnegie-Mellon University): "On the Accurate Long-Time Solution of the Wave Equation in Exterior Domains: Asymptotic Expansions and Corrected Boundary Conditions", ICOMP Report No. 93-4, NASA TM-106117, April 1993, 45 pages.

We consider the solution of scattering problems for the wave equation using approximate boundary conditions at artificial boundaries. These conditions are explicitly viewed as approximations to an exact boundary condition satisfied by the solution on the unbounded domain. We study both the short and long time behavior of the error. It is proved that, in two space dimensions, non-local in time, constant coefficient boundary operator can lead to accurate results uniformly in time for the class of problems we consider. A variable coefficient operator is developed which attains better accuracy (uniformly in time) than is possible with constant coefficient approximations. The theory is illustrated by numerical examples. We also analyze the proposed boundary conditions using energy methods, leading to asymptotically correct error bounds.

Leonard, B. P. (ICOMP); MacVean, M. K. (U. K. Meteorological Office); and Lock, A. P. (U. K. Meteorological Office): "Positivity-Preserving Numerical Schemes for Multidimensional Advection", ICOMP Report No. 93-5, NASA TM-106055, March 1993, 62 pages.

This report describes the construction of an explicit, single time-step, conservative, finite-volume method for multidimensional advective flow, based on a uniformly third-order polynomial interpolation algorithm (UTOPIA). Particular attention is paid to the problem of flow-to-grid angle-dependent, anisotropic distortion typical of 1D schemes used component-wise. The third-order multidimensional scheme automatically includes certain cross-difference terms that guarantee good isotropy (and stability). However, above first-order, polynomial-based advection schemes do not preserve positivity (the multidimensional analogue of monotonicity). For this reason, a multidimensional generalization of the first author's universal flux-limiter is sought. This is a very challenging problem. A simple flux-limiter can be found; but this introduces strong anisotropic distortion. A more sophisticated technique, limiting part of the flux and then restoring the isotropy-maintaining cross-terms afterwards, gives more satisfactory results. Test cases are confined to 2D; 3D extensions are briefly discussed.

Jiang, B.-N. (ICOMP); Lin, T. L. (Livermore Software Technology Corporation); and Povinelli, L. A. (NASA Lewis): "Large-Scale Computation of Incompressible Viscous Flow by Least-Squares Finite Element Method", ICOMP Report No. 93-6, NASA TM-105904, March 1993, 21 pages.

The least-squares finite element method (LSFEM) based on the velocity-pressure-vorticity formulation is applied to large-scale/3D steady incompressible Navier-Stokes problems. This method can accommodate equal-order interpolations, and results in symmetric, positive definite algebraic system which can be solved effectively by simple iterative methods. The first-order velocity-Bernoulli function-vorticity formulation for incompressible viscous flows is also tested. For 3D cases, an additional compatibility equation, i.e., the divergence of vorticity vector should be zero, is included to make the first-order system elliptic. The simple substitution or the Newton's method is employed to linearize the partial differential equations, the LSFEM is used to obtain discretized equations, and the system of algebraic equations is solved using the Jacobi preconditioned conjugate gradient method which avoids formulation of either element or global matrices (matrix-free) to achieve high efficiency. To show the validity of this scheme for large-scale computation, we give numerical results for 2D driven cavity problem at Re = 10000 with 408x400 bilinear elements. The flow in a 3D cavity is calculated at Re = 100, 400, and 1,000 with 50x50x50 trilinear elements. The Taylor-Görtler-like vortices are observed for Re = 1,000.

Liou, William W. (ICOMP); and Shih, Tsan-Hsing (ICOMP): "A Multiple-Scale Model for Compressible Turbulent Flows", ICOMP Report No. 93-7, CMOTT 93-2, NASA TM-106072, March 1993, 22 pages.

A multiple-scale model for compressible turbulent flows is proposed in this paper. It is assumed that turbulent eddy shocklets are formed primarily by the "collisions" of large energetic eddies. The extra straining of the large eddy, due to their interactions with shocklets, enhances the energy cascade to smaller eddies. Model transport equations are developed for the turbulent kinetic energies and the energy transfer rates of the different scale. The turbulent eddy viscosity is determined by the total turbulent kinetic energy and the rate of energy transfer from the large scale to the small scale, which is different from the energy dissipation rate. The model coefficients in the modeled turbulent transport equations depend on the ratio of the turbulent kinetic energy of the large scale to that of the small scale, which renders the model more adaptive to the characteristics of individual flow. The model is tested against compressible free shear layers. The results agree satisfactorily with measurements.

Gajjar, J. S. B. (ICOMP): "Nonlinear Evolution of the First Mode Supersonic Oblique Waves in Compressible Boundary Layers. Part I - Heated/Cooled Walls", ICOMP Report No. 93-8, NASA TM-106087, March 1993, 32 pages.

The nonlinear stability of an oblique mode propagating in a 2D compressible boundary layer is considered under the long wave-length approximation. The growth rate of the wave is assumed to be small so that the ideas of unsteady nonlinear critical layers can be applied. It is shown that the spatial/temporal evolution of the mode is governed by a pair of coupled unsteady nonlinear equations for the disturbance vorticity and density. Expressions for the linear growth rate show clearly the effects of wall heating and cooling, and in particular how heating destablizes the boundary layer for these long wave-length inviscid modes at 0(1) Mach numbers. A generalized expression for the linear growth rate is obtained and is shown to compare very well for a range of frequencies and wave-angles at moderate Mach numbers with full numerical solutions of the linear stability problem. The numerical solution of the nonlinear unsteady critical layer problem using a novel method based on Fourier decomposition and Chebychev collocation is discussed and some results are presented.

Goldstein, Marvin E. (NASA Lewis); and Mathew, Joseph (ICOMP): "The Development of a Mixing Layer Under the Action of Weak Streamwise Vortices", ICOMP Report No. 93-9, NASA TM-106089, March 1993, 36 pages.

The action of weak, streamwise vortices on a plane, incompressible, steady mixing layer is examined in the large-Reynolds-number limit: The outer, inviscid region is bounded by a vortex sheet to which the viscous region is confined. It is shown that the local linear analysis becomes invalid at streamwise distances $0(\varepsilon^{-1})$, where $\varepsilon << 1$ is the crossflow amplitude, and a new nonlinear analysis is constructed for this region. Numerical solutions of the nonlinear problem show that the vortex sheet undergoes an 0(1) change in position and that the solution is ultimately terminated by the appearance of a singularity. The corresponding viscous layer shows downstream thickening, but appears to remain well-behaved up to the singular location.

Ottarsson, Gisli (University of Michigan); and Pierre, Christophe (ICOMP): "A Transfer Matrix Approach to Vibration Localization in Mistuned Blade Assemblies", ICOMP Report No. 93-10, NASA TM 106112, May 1993, 40 pages.

A study of mode localization in mistuned bladed disks is performed using transfer matrices. The transfer matrix approach yields the free response of a general, mono-coupled, perfectly cyclic assembly in closed form. A mistuned structure is represented by random transfer matrices, and the expansion of these matrices in terms of the small mistuning parameter leads to the definition of a measure of sensitivity to mistuning. An approximation of the localization factor, the spatially averaged rate of exponential attenuation per blade-disk sector, is obtained through perturbation techniques in the limits of high and low sensitivity. The methodology is applied to a common model of a bladed disk and the results verified by Monte Carlo simulations. The easily calculated sensitivity measure may prove to be a valuable design tool due to its system-independent quantification of mistuning effects such as mode localization.

Duncan, Beverly S. (Sverdrup); Liou, William W. (ICOMP); and Shih, Tsan-Hsing (ICOMP): "A Multiple-Scale Turbulence Model for Incompressible Flow", ICOMP Report No. 93-11, CMOTT-93-3, NASA TM-106113, January 1993, 14 pages.

A multiple-scale eddy viscosity model is described in this paper. This model splits the energy spectrum into a high wave number regime and a low wave number regime. Dividing the energy spectrum into multiple regimes simplistically emulates the cascade of energy through the turbulence spectrum. The constraints on the model coefficients are determined by examining decaying turbulence and homogeneous turbulence. A direct link between the partitioned energies and the energy transfer process is established through the coefficients. This new model has been calibrated and tested for boundary-free turbulent shear flows. Calculations of mean and turbulent properties show good agreement with experimental data for two mixing layers, a plane jet and a round jet.

Shih, Tsan-Hsing (ICOMP); and Lumley, John L. (Cornell University): "Remarks on Turbulent Constitutive Relations", ICOMP Report No. 93-12, CMOTT-93-06, NASA TM-106116, May 1993, 12 pages.

The paper demonstrates that the concept of turbulent constitutive relations introduced by Lumley (1970) can be used to construct general models for various turbulent correlations. Some of Generalized Cayley-Hamilton formulas for relating tensor products of higher extension to tensor products of lower extension are introduced. The combination of dimensional analysis and invariant theory can lead to "turbulent constitutive relations" (or general turbulence models) for, in principle, any turbulent correlations. As examples, the constitutive relations for Reynolds stresses and scalar fluxes are derived. The results are consistent with ones from RNG theory and two-scale DIA method, but with a more general form.

Zhu, J. (ICOMP); and Shih, T. H. (ICOMP): "Calculations of Turbulent Separated Flows", ICOMP Report No. 93-13, CMOTT-93-05, NASA TM-106154, August 1993, 20 pages.

A numerical study of incompressible turbulent separated flows is carried out by using two-equation turbulence models of the κ-ε type. On the basis of realizability analysis, a new formulation of the eddy-viscosity is proposed which ensures the positiveness of turbulent normal stresses - a realizability condition that most existing two-equation turbulence models are unable to satisfy. The present model is applied to calculate two backward-facing step flows. Calculations with the standard κ-ε model and a recently developed RNG-based κ-ε model are also made for comparison. The calculations are performed with a finite-volume method. A second-order accurate differencing scheme and sufficiently fine grids are used to ensure the numerical accuracy of solutions. The calculated results are compared with the experimental data for both mean and turbulent quantities. The comparison shows that the present model performs quite well for separated flows.

Leonard, B.P. (ICOMP): "Large Time-Step Stability of Explicit One-Dimensional Advection Schemes", ICOMP Report No. 93-14, NASA TM-106203, May 1993, 30 pages.

There is wide-spread belief that most explicit one-dimensional advection schemes need to satisfy the so-called "CFL condition" - that the Courant number, $c = u\Delta t/\Delta x$, must be less than or equal to one, for stability in the von Neumann sense. This puts severe limitations on the time-step in high-speed, fine-grid calculations and is an impetus for the development of implicit schemes, which often require less restrictive time-step conditions for stability, but are more expensive per time-step. However, it turns out that, at least in one dimension, if explicit schemes are formulated in a consistent flux-based conservative fine-volume form, von Neumann stability analysis *does not place any restriction* on the allowable Courant number. Any explicit scheme that is stable for $c < 1$, with a complex amplitude ratio, **G**(c), can be easily extended to arbitrarily large c. The complex amplitude ratio is then given by exp $(-\iota N\Theta)$ $G(\Delta c)$, where N is the integer part of c, and $\Delta c = c - N (<1)$; this is clearly stable. The CFL condition is, in fact, not a stability condition at all, but, rather, a "range restriction" on the "pieces" in a piece-wise polynomial interpolation. When a global view is taken of the interpolation, the need for a CFL condition evaporates. A number of well-known explicit advection schemes are considered and thus extended to large Δt. The analysis also includes a simple interpretation of (large Δt) TVD constraints.

Deissler, Robert J. (ICOMP); Oron, Alexander (ICOMP); and Duh, J. C. (Sverdrup): "Marangoni Instability in a Liquid Layer With Two Free Surfaces", ICOMP Report No. 93-15, NASA TM-106166, July 1993, 26 pages.

We study the onset of the Marangoni instability in a liquid layer with two free nearly insulating surfaces heated from below. Linear stability analysis yields a condition for the emergence of a longwave or a finite wavelength instability from the quiescent equilibrium state. Using the method of asymptotic expansions we derive a weakly nonlinear evolution equation describing the spatiotemporal behavior of the velocity and temperature fields at the onset of the longwave instability. The latter is given by $\Delta M=24$, ΔM being the difference between the upper and the lower Marangoni numbers. It is shown that in some parametric range one convective cell forms across the layer, while in other parametric domains two convective cells emerge between the two free surfaces.

Kao, Kai-Hsiung (ICOMP); Liou, Meng-Sing (NASA Lewis); and Chow, Chuen-Yen (University of Colorado): "Grid Adaptation Using Chimera Composite Overlapping Meshes", ICOMP Report No. 93-16, NASA TM-106163, AIAA-93-3389, May 1993, 34 pages.

The objective of this paper is to perform grid adaptation using composite over-lapping meshes in regions of large gradient to capture the salient features accurately during computation. The Chimera grid scheme, a multiple overset mesh technique, is used in combination with a Navier-Stokes solver. The numerical solution is first converged to a steady state based on an initial coarse mesh. Solution-adaptive enhancement is then performed by using a secondary fine grid system which oversets on top of the base grid in the high-gradient region, but without requiring the mesh boundaries to join in any special way. Communications through boundary interfaces between those separated grids are carried out using tri-linear interpolation. Applications to the Euler equations for shock reflections and to a shock wave/boundary layer interaction problem are tested. With the present method, the salient features are well resolved.

Davis, Dominic A. R. (ICOMP); and Smith, Frank T. (University College, London): "On Nonlinear Tollmien-Schlichting/Vortex Interaction in 3D Boundary Layers", ICOMP Report No. 93-17, NASA TM-106184, May 1993, 29 pages.

The instability of an incompressible 3D boundary layer (that is, one with cross-flow) is considered theoretically and computationally in the context of vortex/wave interactions. Specifically the work centres on two low-amplitude, lower-branch Tollmien-Schlichting waves which mutually interact to induce a weak longitudinal vortex flow; the vortex motion, in turn, gives rise to significant wave-modulation via wall-shear forcing. The characteristic Reynolds number is taken as a large parameter and, as a consequence, the waves' and the vortex motion are governed primarily by triple-deck theory. The nonlinear interaction is captured by a viscous partial-differential system for the vortex coupled with a pair of amplitude equations for each wave pressure. Three distinct possibilities were found to emerge for the nonlinear behavior of the flow solution downstream - an algebraic finite-distance singularity, far-downstream saturation or far-downstream wave-decay (leaving pure vortex flow) - depending on the input conditions, the wave angles and the size of the cross-flow.

Zhu, J. (ICOMP); and Shih, Tsan-Hsing (ICOMP): "A Numerical Study of Confined Turbulent Jets", ICOMP Report No. 93-18, CMOTT 93-07, NASA TM-106197, June 1993, 27 pages.

A numerical investigation is reported of turbulent incompressible jets confined in two ducts, one cylindrical and the other conical with a 5° divergence. In each case, three Craya-Curtet numbers are considered which correspond, respectively, to flow situations with no, moderate and strong recirculation. Turbulence closure is achieved by using the κ-ε model and a recently proposed realizable Reynolds stress algebraic equation model that relates the Reynolds stresses explicitly to the quadratic terms of the mean velocity gradients and ensures the positiveness of each component of the turbulent kinetic energy. Calculations are carried out with a finite-volume procedure using boundary-fitted curvilinear co-ordinates. A second-order accurate, bounded convection scheme and sufficiently fine grids are used to prevent the solutions from being contaminated by numerical diffusion. The calculated results are compared extensively with the available experimental data. It is shown that the numerical methods presented are capable of capturing the essential flow features observed in the experiments and that the realizable Reynolds stress algebraic equation model performs much better than the κ-ε model for this class of flows of great practical importance.

Bochev, Pavel B. (Virginia Polytechnic Institute and State University); and Gunzburger, Max D. (ICOMP): "Accuracy of Least-Squares Methods for the Navier-Stokes Equations", ICOMP Report No. 93-19, NASA TM-106209, June 1993, 22 pages.

Recently there has been substantial interest in least-squares finite element methods for velocity-vorticity-pressure formulations of the incompressible Navier-Stokes equations. The main cause for this interest is the fact that algorithms for the resulting discrete equations can be devised which require the solution of only symmetric, positive definite systems of algebraic equations. On the other hand, it is well-documented that methods using the vorticity as a primary variable often yield very poor approximations. Thus, here we study the accuracy of these methods through a series of computational experiments, and also comment on theoretical error estimates. It is found, despite the failure of standard methods for deriving error estimates, that computational evidence suggests that these methods are, at the least, nearly optimally accurate. Thus, in addition to the desirable matrix properties yielded by least-squares methods, one also obtains accurate approximations.

Shih, Tsan-Hsing (ICOMP); Liou, William (ICOMP); Shabbir, Aamir (ICOMP); Yang, Zhigang (ICOMP); and Zhu, Jiang (ICOMP): "A Vorticity Dynamics Based Model for the Turbulent Dissipation—Model Development and Validation", ICOMP Report No. 93-20, CMOTT 93-08, NASA TM 106177, January 1994.

A new model dissipation rate equation is proposed based on the dynamic equation of the mean-square vorticity fluctuation for large Reynolds number turbulence. The advantage of working with the vorticity fluctuation equation is that the physical meanings of the terms in this equation are more clear than those in the dissipation rate equation. Hence, the model development based on the vorticity fluctuation equation is more straightforward. The resulting form of the model equation is consistent with the spectral energy cascade analysis introduced by Lumley (1992). The proposed model dissipation rate equation is numerically well behaved and can be applied to any level of turbulence modeling. In the present study, it is applied to a realizable eddy viscosity model. Flows that are examined include: (i) rotating homogeneous shear flows; (ii) free shear flows; (iii) a channel flow and flat plate boundary layers with and without pressure gradients; and (iv) backward facing step separated flows. In most cases, the present model predictions show considerable improvement over the standard κ-ε model.

Steinthorsson, Erlendur (ICOMP); Liou, Meng-Sing (NASA Lewis); Povinelli, Louis A. (NASA Lewis); and Arnone, Andrea (ICOMP): "Numerical Simulations of 3D Laminar Flow Over a Backward Facing Step; Flow Near Side Walls", ICOMP Report No. 93-21, NASA TM-106248, July 1993, 10 pages.

This paper reports the results of numerical simulations of steady, laminar flow over a backward-facing step. The governing equations used in the simulations are the full "compressible" Navier-Stokes equations, solutions to which were computed by using a cell-centered, finite volume discretization. The convection terms of the governing equations were discretized by using the Advection Upwind Splitting Method (AUSM), whereas the diffusion terms were discretized using central differencing formulas. The validity and accuracy of the numerical solutions were verified by comparing the results to existing experimental data for flow at identical Reynolds numbers in the same backstep geometry. The paper focuses attention on the details of the flow field near the side wall of the geometry.

Brereton, G. J. (ICOMP); and Mankbadi, R. R. (NASA Lewis): "A Rapid-Distortion-Theory Turbulence Model for Developed Unsteady Wall-Bounded Flow", ICOMP Report No. 93-22, NASA TM-106249, July 1993, 20 pages.

A new approach to turbulence modeling in unsteady developed flows has recently been introduced [1], based on results of rapid distortion theory. The approach involves closing the κ-ι equations for the organized unsteady component of the flow by modeling local unsteadiness as a rapid distortion of the local structure of the parent turbulent flow, in terms of an effective strain parameter α_{eff} [2]. In this paper, the phase-conditioned equations of motion are developed to accommodate a new unsteady dissipation model and local effects of the slow-relaxation time scale of the parent flow. The model equations are tested against measurements of the response of a fully-developed turbulent pipe flow to the superposition of sinusoidal streamwise oscillation. Good agreement is found between measurements and predictions over a wide range of frequencies of unsteadiness, indicating that this approach may be particularly well suited to modeling of unsteady turbulent flows which are perturbations about a well characterized mean.

Winterscheidt, Daniel L. (ICOMP): "A p-Version Finite Element Method for Steady Incompressible Fluid Flow and Convective Heat Transfer", ICOMP Report No. 93-23, NASA TM-106260, July 1993, 26 pages.

A new p-version finite element formulation for steady, incompressible fluid flow and convective heat transfer problems is presented. The steady-state residual equations are obtained by considering a limiting case of the least-squares formulation for the transient problem. The method circumvents the Babuska-Brezzi condition, permitting the use of equal-order interpolation for velocity and pressure, without requiring the use of arbitrary parameters. Numerical results are presented to demonstrate the accuracy and generality of the method.

Yang, Z. (ICOMP); and Shih, Tsan-Hsing (ICOMP): "A Galilean and Tensorial Invariant κ-ε Model for Near Wall Turbulence", ICOMP Report No. 93-24, CMOTT 93-10, NASA TM-106263, AIAA-93-3105, July 1993, 12 pages.

A κ-ε model is proposed for wall bounded turbulent flows. In this model, the eddy viscosity is characterized by a turbulent velocity scale and a turbulent time scale. The time scale is bounded from below by the Kolmogorov time scale. The dissipation rate equation is reformulated using this time scale and no singularity exists at the wall. A new parameter $R = \frac{\kappa}{s\upsilon}$ is introduced to characterize the damping function in the eddy viscosity. This parameter is determined by local properties of both the mean and the turbulent flow fields and is free from any geometry parameter. The proposed model is then Galilean and tensorial invariant. The model constraints used are the same as in the high Reynolds number Standard κ-ε Model. Thus, the proposed model will be also suitable for flows far from the wall. Turbulent channel flows and turbulent boundary layer flows with and without pressure gradients are calculated. Comparisons with the data from direct numerical simulations and experiments show that the model predictions are excellent for turbulent channel flows and turbulent boundary layers with favorable pressure gradients, good for turbulent boundary layers with zero pressure gradients, and fair for turbulent boundary layers with adverse pressure gradients.

Arnone, Andrea (ICOMP): "Viscous Analysis of 3D Rotor Flows Using a Multigrid Method", ICOMP Report No. 93-25, NASA TM 106266, July 1993, 36 pages.

A 3D code for rotating blade-row flow analysis has been developed. The space discretization uses a cell-centered scheme with eigenvalues scaling for the artificial dissipation. The computational efficiency of a four-stage Runge-Kutta scheme is enhanced by using variable coefficients, implicit residual smoothing, and a full-multigrid method. An application is presented for the NASA rotor 67 transonic fan. Due to the blade stagger and twist, a zonal, non-periodic H-type grid is used to minimize the mesh skewness. The calculation is validated by comparing it with experiments in the range from the maximum flow rate to a near-stall condition. A detailed study of the flow structure near peak efficiency and near-stall is presented by means of pressure distribution and particle traces inside boundary layers.

Hayder, M. Ehtesham (ICOMP); and Turkel, Eli (ICOMP): "High Order Accurate Solutions of Viscous Problems", ICOMP Report No. 93-26, NASA TM 106267, AIAA-93-3074, July 1993, 14 pages.

We consider a fourth order extension to MacCormack's scheme. The original extension was fourth order only for the inviscid terms but was second order for the viscous terms. We show how to modify the viscous terms so that the scheme is uniformly fourth order in the spatial derivatives. Applications are given to some boundary layer flows. In addition, for applications to shear flows the effect of the outflow boundary conditions are very important. We compare the accuracy of several of these different boundary conditions for both boundary layer and shear flows. Stretching at the outflow usually increases the oscillations in the numerical solution but the addition of a filtered sponge layer (with or without stretching) reduces such oscillations. The oscillations are generated by insufficient resolution of the shear layer. When the shear layer is sufficiently resolved then oscillations are not generated and there is less of a need for a nonreflecting boundary condition.

Afolabi, Dare (ICOMP); and Mehmed, Oral (NASA Lewis): "On Curve Veering and Flutter of Rotating Blades", ICOMP Report No. 93-27, NASA TM 106282, August 1993, 21 pages.

The eigenvalues of rotating blades usually change with rotation speed according to the Stodola-Southwell criterion. Under certain circumstances, the loci of eigenvalues belonging to two distinct modes of vibration approach each other very closely, and it may appear as if the loci cross each other. However, our study indicates that the observable frequency loci of an *undamped* rotating blade do not cross, but must either repel each other (leading to "curve veering"), or attract each other (leading to "frequency coalescence"). Our results are reached by using standard arguments from algebraic geometry - the theory of algebraic curves and catastrophe theory. We conclude that it is important to resolve an apparent crossing of eigenvalue loci into either a frequency coalescence or a curve veering, because frequency coalescence is dangerous since it leads to flutter, whereas curve veering does not precipitate flutter and is, therefore, harmless with respect to elastic stability.

Liou, William W. (ICOMP): "Linear Instability of Curved Free Shear Layers", ICOMP Report No. 93-28, CMOTT 93-11, NASA TM 106290, AIAA 93-03252, July 1993, 18 pages.

The linear inviscid hydrodynamic stability of slightly curved free mixing layers is studied in this paper. The disturbance equation is solved numerically using a shooting technique. Two mean velocity profiles that represent stably and unstably curved free mixing layers are considered. Results are shown for cases of five curvature Richardson numbers. The stability characteristics of the shear layer are found to vary significantly with the introduction of the curvature effects. The results also indicate that, in a manner similar to the Gortler vortices observed in a boundary layer along a concave wall, instability modes of spatially developing streamwise vortex pairs may appear in centrifugally unstable curved mixing layers.

Demuren, A. O. (ICOMP) and Ibraheem, S. O. (Old Dominion University): "On The Stability Analysis of Approximate Factorization Methods for 3D Euler and Navier-Stokes Equations", ICOMP Report No. 93-29, NASA TM 106314, October 1993, 38 pages.

The convergence characteristics of various approximate factorizations for the 3D Euler and Navier-Stokes equations are examined using the von-Neumann stability analysis method. Three upwind-difference based factorizations and several central-difference based factorizations are considered for the Euler equations. In the upwind factorizations both the flux-vector splitting methods of Steger and Warming and van Leer are considered. Analysis of the Navier-Stokes equations is performed only on the Beam and Warming central-difference scheme. The range of CFL numbers over which each factorization is stable is presented for one-, two- and three-dimensional flow. Also presented for each factorization is the CFL number at which the maximum eigenvalue is minimized, for all Fourier components, as well as for the high frequency range only. The latter is useful for predicting the effectiveness of multigrid procedures with these schemes as smoothers. Further, local mode analysis is performed to test the suitability of using a uniform flow field in the stability analysis. Some inconsistencies in the results from previous analyses are resolved.

Gunzburger, Max D. (ICOMP); and Lee, Hyung C. (Virginia Polytechnic Institute and State University): "Analysis, Approximation, and Computation of a Coupled Solid/Fluid Temperature Control Problem", ICOMP Report 93-30, NASA TM 106351, September 1993, 34 pages.

An optimization problem is formulated motivated by the desire to remove temperature peaks, i.e., "hot spots", along the bounding surfaces of containers of fluid flows. The heat equation of the solid container is coupled to the energy equations for the fluid. Heat sources can be located in the solid body, the fluid, or both. Control is effected by adjustments to the temperature of the fluid at the inflow boundary. Both mathematical analyses and computational experiments are given.

Jiang, Bo-Nan (ICOMP); Hou, Lin-Jun (ICOMP); and Lin, Tsung-Liang (Livermore Software Technology Corporation): "Least-Squares Finite Element Solutions for Three-Dimensional Backward-Facing Step Flow", ICOMP Report 93-31, NASA TM 106353, August 1993, 22 pages.

Comprehensive numerical solutions of the steady state incompressible viscous flow over a three-dimensional backward-facing step up to *Re* = 800 are presented. The results are obtained by the least-squares finite element method (LSFEM) which is based on the velocity-pressure-vorticity formulation. The computed model is of the same size as that of Armaly's experiment. Three-dimensional phenomena are observed even at low Reynolds number. The calculated values of the primary reattachment length are in good agreement with experimental results.

Yungster, Shaye (ICOMP); and Rabinowitz, Martin J. (NASA Lewis): "Numerical Study of Shock-Induced Combustion in Methane-Air Mixtures", ICOMP Report 93-32, NASA TM 106354, AIAA 93-1917, June 1993, 14 pages.

The shock-induced combustion of methane-air mixtures in hypersonic flows is investigated using a new reaction mechanism consisting of 19 reacting species and 52 elementary reactions. This reduced model is derived from a full kinetic mechanism via the Detailed Reduction technique. Zero-dimensional computations of several shock-tube experiments are presented first. The reaction mechanism is then combined with a fully implicit Navier-Stokes CFD code to conduct numerical simulations of two-dimensional and axisymmetric shock-induced combustion experiments of stoichiometric methane-air mixtures at a Mach number of M = 6.61. Applications to the ram accelerator concept are also presented.

Van Der Wegt, J. J. W. (ICOMP): "Higher-Order Accurate Osher Schemes with Application to Compressible Boundary Layer Stability", ICOMP Report No. 93-33, CMOTT-93-12, NASA TM 106355, AIAA 93-3051, July 1993, 16 pages.

Two fourth order accurate Osher schemes are presented which maintain higher order accuracy on nonuniform grids. They use either a conservative finite difference or finite volume discretization. Both methods are successfully used for direct numerical simulations of flat plate boundary layer instability at different Mach numbers. Results of growth rates of Tollmien-Schlichting waves compare well with direct simulations of incompressible flow and for compressible flow with results obtained by solving the parabolic stability equations.

Steinthorsson E. (ICOMP); Liou M. S.(NASA Lewis); and Povinelli, L. A. (NASA Lewis): "Development of an Explicit Multiblock/Multigrid Flow Solver for Viscous Flows in Complex Geometries", ICOMP Report 93-34, NASA TM 106356, AIAA 93-2380, June 1993, 16 pages.

A new computer program is being developed for doing accurate simulations of compressible viscous flows in complex geometries. The code employs the full compressible Navier-Stokes equations. The eddy viscosity model of Baldwin and Lomax is used to model the effects of turbulence on the flow. A cell centered finite volume discretization is used for all terms in the governing equations. The Advection Upwind Splitting Method (AUSM) is used to compute the inviscid fluxes, while central differencing is used for the diffusive fluxes. A four-stage Runge-Kutta time integration scheme is used to march solutions to steady state, while convergence is enhanced by a multigrid scheme, local time-stepping and implicit residual smoothing. To enable simulations of flows in complex geometries, the code uses composite structured grid systems where all grid lines are continuous at block boundaries (multiblock grids). Example results are shown for a flow in a linear cascade, a flow around a circular pin extending between the main walls in a high aspect-ratio channel, and a flow of air in a radial turbine coolant passage.

Mallier, R. (McGill University); and Maslowe, S. A. (ICOMP): "Parametric Resonant Triad Interactions in a Free Shear Layer", ICOMP Report 93-35, NASA TM 106365, October 1993, 22 pages.

We investigate the weakly nonlinear evolution of a triad of nearly-neutral modes superimposed on a mixing layer with velocity profile $\overline{u} = u_m + \tanh y$. The perturbation consists of a plane wave and a pair of oblique waves each inclined at approximately 60° to the mean flow direction. Because the evolution occurs on a relatively fast time scale, the critical layer dynamics dominate the process and the amplitude evolution of the oblique waves is governed by an integro-differential equation. The long-time solution of this equation predicts very rapid (exponential of an exponential) amplification and we discuss the pertinence of this result to vortex pairing phenomena in mixing layers.

Liu, Chaoqun (ICOMP); and Liu, Zhining (University of Colorado, Denver): "Multigrid Direct Numerical Simulation of the Whole Process of Flow Transition in 3-D Boundary Layers", ICOMP Report 93-36, CMOTT-93-13, NASA TM 106369, November 1993, 40 pages.

A new technology was developed in this study which provides a successful numerical simulation of the whole process of flow transition in 3-D boundary layers, including linear growth, secondary instability, breakdown, and transition at relatively low CPU cost. Most other spatial numerical simulations require high CPU cost and blow up at the stage of flow breakdown. A fourth-order finite difference scheme on stretched and staggered grids, a fully implicit time-marching technique, a semi-coarsening multigrid based on the so-called approximate line-box relaxation, and a buffer domain for the outflow boundary conditions were all used for high-order accuracy, good stability, and fast convergence. A new fine-coarse-fine grid mapping technique was developed to keep the code running after the laminar flow breaks down. The computational results are in good agreement with linear stability theory, secondary instability theory, and some experiments. The cost for typical case with 162 x 34 x 34 grid is around 2 CRAY-YMP CPU hours for 10 T-S periods.

Arnone, Andrea (ICOMP); Liou, Meng-Sing (NASA Lewis); and Povinelli, Louis A. (NASA Lewis): "Multigrid Time-Accurate Integration of Navier-Stokes Equations", ICOMP Report 93-37, NASA TM 106373, November 1993, 10 pages.

Efficient acceleration techniques typical of explicit steady-state solvers are extended to time-accurate calculations. Stability restrictions are greatly reduced by means of fully implicit time discretization. A four-stage Runge-Kutta scheme with local time stepping, residual smoothing, and multigridding is used instead of traditional time-expensive factorizations. Some applications to natural and forced unsteady viscous flows show the capability of the procedure.

Hagstrom, Thomas (ICOMP); and Lorenz, Jens (University of New Mexico): "Boundary Conditions and the Simulation of Low Mach Number Flows", ICOMP Report 93-38, NASA TM 106374, November 1993, 16 pages.

We consider the problem of accurately computing low Mach number flows, with the specific intent of studying the interaction of sound waves with incompressible flow structures, such as concentrations of vorticity. This is a multiple time (and/or space) scales problem, leading to various difficulties in the design of numerical methods. In this paper, we concentrate on one of these difficulties—the development of boundary conditions at artificial boundaries which allow sound waves and vortices to radiate to the far field. Nonlinear model equations are derived based on assumptions about the scaling of the variables. We then linearize these about a uniform flow and systematically derive exact boundary conditions using transform methods. Finally, we compute useful approximations to the exact conditions which are valid for small Mach number and small viscosity.

Deissler, Robert (ICOMP): "Thermally-Sustained Structure in Convectively Unstable Systems", ICOMP Report 93-39, NASA TM 106375, November 1993, 16 pages.

The complex Ginzburg-Landau equation with a thermal noise term is studied under conditions when the system is convectively unstable. Under these conditions, the noise is selectively and spatially amplified giving rise to a noise-sustained structure. Analytical results, applicable to a wide range of physical systems, are derived for the variance, and the coefficients and thermal noise term are determined for Taylor-Couette flow with an axial through-flow. Comparison is made to recent experiments.

Zhu, J. (ICOMP); and Shih, T. H. (ICOMP): "Computation of Confined Coflow Jets With Three Turbulence Models", ICOMP Report 93-40, CMOTT-93-14, NASA TM 106378, AIAA-93-3120, July 1993, 12 pages.

A numerical study of confined jets in a cylindrical duct is carried out to examine the performance of two recently proposed turbulence models: an RNG-based κ-ε model and a realizable Reynolds stress algebraic equation model. The former is of the same form as the standard κ-ε model but has different model coefficients. The latter uses an explicit quadratic stress-strain relationship to model the turbulent stresses and is capable of ensuring the positivity of each turbulent normal stress. The flow considered involves recirculation with unfixed separation and reattachment points and severe adverse pressure gradients, thereby providing a valuable test of the predictive capability of the models for complex flows. Calculations are performed with a finite-volume procedure. Numerical credibility of the solutions is ensured by using second-order accurate differencing schemes and sufficiently fine grids. Calculations with the standard κ-ε model are also made for comparison. Detailed comparisons with experiments show that the realizable Reynolds stress algebraic equation model consistently works better than does the standard κ-ε model in capturing the essential flow features, while the RNG-based κ-ε model does not seem to give improvements over the standard κ-ε model under the flow conditions considered.

Rubinstein, Robert (ICOMP): "Time Dependent Turbulence Modeling and Analytical Theories of Turbulence", ICOMP Report 93-41, CMOTT-93-15, NASA TM 106379, November 1993, 28 pages.

By simplifying the direct interaction approximation (DIA) for turbulent shear flow, time dependent formulas are derived for the Reynolds stresses which can be included in two equation models. The Green's function is treated phenomenologically, however following Smith and Yakhot, we insist on the short and long time limits required by DIA. For small strain rates, perturbative evaluation of the correlation function yields a time dependent theory which includes normal stress effects in simple shear flows. From this standpoint, the phenomenological Launder-Reece-Rodi model is obtained by replacing the Green's function by its long time limit. Eddy damping corrections to short time behavior initiate too quickly in this model; in contrast, the present theory exhibits strong suppression of eddy damping at short times. A time dependent theory for large strain rates is proposed in which large scales are governed by rapid distortion theory while small scales are governed by Kolmogorov inertial range dynamics. At short times and large strain rates, the theory closely matches rapid distortion theory, but at long times it relaxes to an eddy damping model.

Pletcher, R. H. (ICOMP); and Chen, K.-H. (Ohio Aerospace Institute): "On Solving the Compressible Navier-Stokes Equations for Unsteady Flows at Very Low Mach Numbers", ICOMP Report 93-42, NASA TM 106380, AIAA-93-3368, July 1993, 12 pages.

The properties of a preconditioned, coupled, strongly implicit finite-difference scheme for solving the compressible Navier-Stokes equations in primitive variables are investigated for two unsteady flows at low speeds, namely the impulsively started driven cavity and the startup of pipe flow. For the shear-driven cavity flow, the computational effort was observed to be nearly independent of Mach number, especially at the low end of the range considered. This Mach number independence was also observed for steady pipe flow calculations; however, rather different conclusions were drawn for the unsteady calculations. In the pressure-driven pipe startup problem, the compressibility of the fluid began to significantly influence the physics of the flow development at quite low Mach numbers. The present scheme was observed to produce the expected characteristics of completely incompressible flow when the Mach number was set at very low values. Good agreement with incompressible results available in the literature was observed.

Afolabi, D. (ICOMP): "Flutter Analysis Using Transversality Theory", ICOMP Report 93-43, NASA TM 106382, November 1993, 24 pages.

A new method of calculating flutter boundaries of undamped aeronautical structures is presented. The method is an application of the weak transversality theorem used in catastrophe theory. In the first instance, the flutter problem is cast in matrix form using a frequency domain method, leading to an eigenvalue matrix. The characteristic polynomial resulting from this matrix usually has a smooth dependence on the system's parameters. As these parameters change with operating conditions, certain critical values are reached at which flutter sets in. Our approach is to use the transversality theorem in locating such flutter boundaries using this criterion: *at a flutter boundary, the characteristic polynomial does not intersect the axis of the abscissa transversally.* Formulas for computing the flutter boundaries and flutter frequencies of structures with two degrees of freedom are presented, and extension to multi degree of freedom systems is indicated. The formulas have obvious applications in, for instance, problems of panel flutter at supersonic Mach numbers.

Liou, William W. (EDITOR): "Center for Modeling of Turbulence and Transition: Research Briefs—1993", ICOMP Report 93-44, NASA TM 106383, CMOTT-93-16, February 1994, 251 pages.

This research brief contains the progress reports of the research staff of the Center for Modeling of Turbulence and Transition (CMOTT) from June 1992 to July 1993. It is also an annual report to the Institute for Computational Mechanics in Propulsion located at the Ohio Aerospace Institute and NASA Lewis Research Center. The main objectives of the research activities at CMOTT are to develop, validate and implement turbulence and transition models for flows of interest in propulsion systems. Currently, our research covers eddy viscosity one- and two-equation models, Reynolds-stress algebraic equation models, Reynolds-stress transport equation models, nonequilibrium multiple-scale models, bypass transition models, joint scalar probability density function models and Renormalization Group Theory and Direct Interaction Approximation methods. Some numerical simulations (LES and DNS) have also been carried out to support the development of turbulence modeling. Last year was CMOTT's third year in operation. During this period, in addition to the above mentioned research, CMOTT has also hosted the following programs: an eighteen-hour short course on "Turbulence—Fundamentals and Computational Modeling (Part I)" given by CMOTT at the NASA Lewis Research Center; a productive summer visitor research program that has generated many encouraging results; collaborative programs with industry customers (e.g. P.&W. and RocketDyne) to help improve their turbulent flow calculations for propulsion system designs; a biweekly CMOTT seminar series with speakers from within and without NASA Lewis Research Center, including foreign speakers. In addition, CMOTT members have been actively involved in the national and international turbulence research activities. The current CMOTT roster and organization are listed in Appendix A. Listed in Appendix B are the abstracts of the biweekly CMOTT seminars. Appendix C lists the papers contributed by CMOTT members.

Nobari, M. R. (University of Michigan); and Jan, Y.-J. (University of Michigan); and Tryggvason, G. (ICOMP): "Head-On Collision of Drops — A Numerical Investigation", ICOMP Report 93-45, NASA TM 106394, November 1993, 50 pages.

The head-on collision of equal sized drops is studied by full numerical simulations. The Navier-Stokes equations are solved for the fluid motion both inside and outside the drops using a front tracking/finite difference technique. The drops are accelerated toward each other by a body force that is turned off before the drops collide. When the drops collide, the fluid between them is pushed outward leaving a thin layer bounded by the drop surface. This layer gets progressively thinner as the drops continue to deform and in several of our calculations we artificially remove this double layer once it is thin enough, thus modeling rupture. If no rupture takes place, the drops always rebound, but if the film is ruptured the drops may coalesce permanently or coalesce temporarily and then split again.

Shabbir, A. (ICOMP); Shih, T.-H. (ICOMP); and Povinelli, L. A. (NASA Lewis) Compilers: "Workshop on Computational Turbulence Modeling", ICOMP Report 93-46, NASA CP 10130, February 1994, 442 pages.

The purpose of this meeting was to discuss the current status and future development of turbulence modeling in computational fluid dynamics for aerospace propulsion systems. Various turbulence models have been developed and applied to different turbulent flows over the past several decades and it is becoming more and more urgent to assess their performance in various complex situations. In order to help users in selecting and implementing appropriate models in their engineering calculations, it is important to identify the capabilities as well as the deficiencies of these models. This also benefits turbulence modelers by permitting them to further improve upon the existing models. This workshop was designed for exchanging ideas and enhancing collaboration between different groups in the Lewis community who are using turbulence models in propulsion related CFD. In this respect this workshop will help the Lewis goal of excelling in propulsion related research. This meeting had seven sessions for presentations and one panel discussion over a period of 2 days. Each presentation session was assigned to one or two branches (or groups) to present their turbulence related research work. Each group was asked to address at least the following points: current status of turbulence model applications and developments in the research; progress and existing problems; requests about turbulence modeling. The panel discussion session was designated for organizing committee members to answer management and technical questions from the audience and to make concluding remarks.

Leonard, B. P. (ICOMP): "Order of Accuracy of QUICK and Related Convection-Diffusion Schemes", ICOMP Report 93-47, NASA TM 106402, November 1993, 32 pages.

This report attempts to correct some misunderstandings that have appeared in the literature concerning the order of accuracy of the QUICK scheme for steady-state convective modelling. Other related convection-diffusion schemes are also considered. The original one-dimensional QUICK scheme written in terms of *nodal-point* values of the convected variable (with a 1/8-factor multiplying the "curvature" term) is indeed a third-order representation of the finite-*volume* formulation of the convection *operator average* across the control volume, written naturally in flux-difference form. An alternative single-point upwind difference scheme (SPUDS) using node values (with a 1/6-factor) is a third-order representation of the finite *difference single-point* formulation; this can be written in a *pseudo*-flux-difference form. These are both third-order convection schemes; however, the QUICK finite-volume convection operator is 33 percent more accurate than the single-point implementation of SPUDS. Another finite-volume scheme, writing convective fluxes in terms of *cell-average* values, requires a 1/6-factor for third-order accuracy. For completeness, one can also write a single-point formulation of the convective derivative in terms of cell averages, and then express this in pseudo-flux-difference form; for third-order accuracy, this requires a curvature factor of 5/24. Diffusion operators are also considered in both single-point and finite-volume formulations. Finite-volume formulations are found to be significantly more accurate. For example, classical second-order central differencing for the second derivative is exactly twice as accurate in a finite-volume formulation as it is in single-point.

Jorgenson, Philip (NASA Lewis); and Pletcher, Richard (ICOMP): "An Implicit Numerical Scheme for the Simulation of Internal Viscous Flows on Unstructured Grids", ICOMP Report 93-48, NASA TM 106437, AIAA 94-0306, January 1994, 20 pages.

The Navier-Stokes equations are solved numerically for two-dimensional steady viscous laminar flows. The grids are generated based on the method of Delaunay triangulation. A finite-volume approach is used to discretize the conservation law form of the compressible flow equations written in terms of primitive variables. A preconditioning matrix is added to the equations so that low Mach number flows can be solved economically. The equations are time marched using either an implicit Gauss-Seidel iterative procedure or a solver based on a conjugate gradient like method. A four color scheme is employed to vectorize the block Gauss-Seidel relaxation procedure. This increases the memory requirements minimally and decreases the computer time spent solving the resulting system of equations substantially. A factor of 7.6 speedup in the matrix solver is typical for the viscous equations. Numerical results are obtained for inviscid flow over a bump in a channel at subsonic and transonic conditions for validation with structured solvers. Viscous results are computed for developing flow in a channel, a symmetric sudden expansion, periodic tandem cylinders in a cross-flow, and a four-port valve. Comparisons are made with available results obtained by other investigators.

Ajmani, Kumud (ICOMP); Liou, Meng-Sing (NASA Lewis); and Rodger W. Dyson (NASA Lewis): "Preconditioned Implicit Solvers for the Navier-Stokes Equations on Distributed-Memory Machines", ICOMP Report 93-49, NASA TM 106449, AIAA 94-0408, January 1994, 13 pages.

The GMRES method is parallelized, and combined with local preconditioning to construct an implicit parallel solver to obtain steady-state solutions for the Navier-Stokes equations of fluid flow on distributed-memory machines. The new implicit parallel solver is designed to preserve the convergence rate of the equivalent 'serial' solver. A static domain-decomposition is used to partition the computational domain amongst the available processing nodes of the parallel machine. The SPMD (Single-Program Multiple-Data) programming model is combined with message-passing tools to develop the parallel code on a 32-node Intel Hypercube and a 512-node Intel Delta machine. The implicit parallel solver is validated for internal and external flow problems, and is found to compare identically with flow solutions obtained on a Cray Y-MP/8. The computational speed on 32 processing nodes of the i860 machines is comparable to the speed on a single processor of the Cray Y-MP. A peak computational speed of 2300 MFlops/sec has been achieved on 512 nodes of the Intel Delta machine, for a problem size of 1024K equations (256K grid points).

Wada, Yasuhiro (ICOMP); and Liou, Meng-Sing (NASA Lewis): "A Flux Splitting Scheme With High-Resolution and Robustness for Discontinuities", ICOMP Report 93-50, NASA TM 106452, AIAA 94-0083, January 1994, 24 pages.

A flux splitting scheme is proposed for the general non-equilibrium flow equations with an aim at removing numerical dissipation of Van-Leer-type flux-vector splittings on a contact discontinuity. The scheme obtained is also recognized as an improved Advection Upwind Splitting Method (AUSM) where a slight numerical overshoot immediately behind the shock is eliminated. The proposed scheme has favorable properties: high-resolution for contact discontinuities; conservation of enthalpy for steady flows; numerical efficiency; applicability to chemically reacting flows. In fact, for a single contact discontinuity, even if it is moving, this scheme gives the numerical flux of the exact solution of the Riemann problem. Various numerical experiments including that of a thermo-chemical nonequilibrium flow were performed, which indicate no oscillation and robustness of the scheme for shock/expansion waves. A cure for carbuncle phenomenon is discussed as well.

Shih, Tsan-Hsing (ICOMP); Shabbir, Aamir (ICOMP); and Lumley, John L. (Cornell University): "Realizability in Second Moment Turbulence Closures Revisited", ICOMP Report 93-51, CMOTT-93-18, NASA TM 106469, January 1994, 68 pages.

Realizability in turbulence modeling, i.e., the non-negativity of the normal Reynolds stresses and scalar variances and the Schwarz' inequality between any two fluctuating quantities, has been accepted as an important concept in the study of turbulence closures (Schumann 1977, Lumley 1978, Shih and Lumley 1985). Properly developed model equations based on realizability can prevent non-physical results and also reduce numerical stiffness. However, much confusion about the realizability concept has been created recently by Speziale et al. (1993). In order to provide a better understanding of realizability and its application in turbulence modeling, this paper discusses the following important subjects: the properties of general turbulent correlation matrices and the related realizability concept, the general realizability conditions and the method of their implementation. We also present the detailed realizability constraints for various model terms in the second moment equations (including Reynolds stresses and scalar fluxes) that, we believe, are useful for developing more general turbulence models. As an example, we demonstrate the realizable behavior of such realizably constrained second moment closures based on the work of Shih and Lumley (1985) for turbulent flows under various critical situations. For the purpose of comparisons, a recently proposed model by Speziale et al. (1991) that makes no attempt to satisfy realizability and the IP model of Launder, Reece and Rodi (1975) are also included. Comparisons clearly show the superior realizable behavior of the models based on realizability over that of the models not considering realizability.

SEMINARS

(* = CMOTT Seminars)
(= Computational Aeroacoustics (CAA) Seminars)**

***Ameri, Ali (NASA Lewis):** "Some Turbulence Modeling Issues Related to Heat Transfer in Turbomachinery"

Navier-Stokes calculations were carried out in order to predict the heat transfer rates on surfaces of turbine blades. The code TRAF was modified to handle a variety of two-equation models in addition to the baseline Baldwin and Lomax model. The calculations were performed efficiently by utilizing a multigrid method. The results of the calculations generally agree with the experimental measurements in the laminar and the fully turbulent regimes. The transition process is, however, not well predicted. The two-equation model results also show a very distinct sensitivity to the assigned free stream length scale of turbulence. The accuracy level of the results obtained using the Baldwin-Lomax model was comparable to those obtained using the two-equation models. With that in mind, a transition model was incorporated into the code which, when used in conjunction with the B-L model, produced very good results. The extension of the two-equation and the transition model to 3D calculations will be addressed and some example cases will be considered.

***Anderson, Bernhard H. (NASA Lewis); and Kapoor, Kamlesh (NASA Lewis):** "A Study on Bifurcated Transitioning S-Ducts for High Speed Inlet Applications"

Mission factors for the next generation of supersonic cruise aircraft are driving designers to very short two-dimensional bifurcated inlet systems highly integrated into the airframe. The ATSF (Avion de Transport Supersonique du Futur) is one such highly integrated, highly efficient engine airframe system. This requirement is moving design engineers to consider diffusers which operate along the stall line, i.e., diffuser operates very close to flow separation over most of the duct length. This is the most efficient operating point where turbulence models tend to breakdown and where artificial viscosity (or diffusivity) may seriously corrupt the physical process and can even effect inlet design parameters.

This study explores the flow physics of bifurcated transitioning S-ducts operating along the stall line through a comparative study of Reduced Navier-Stokes and Full Navier-Stokes analyses, documenting where the two solutions agree and where they disagree. The purpose of this study is to understand the issues of analyzing such systems and to design a benchmark experiment for CFD code and turbulence model validation. The authors invite anyone in performing a prediction <u>before</u> experimental data is available, and to help in the design of the experiment itself.

***Armstrong, Rob (Sandia National Labs):** "A Frame-Based Turbulent Reacting Flow Application for Parallel Computers"

Chemically reacting flow models, as implemented on serial (vector) machines, are currently limited in size and realism by the speed of the machines. Though turbulent flow problems alone are computationally intensive, the computation for detailed chemistry in turbulent reacting flow models dominates the fluid mechanics. Thus chemically reacting flows are an attractive candidate for massively parallel and distributed systems. We have created such a model using an object-oriented frame-based toolkit.

Currently, there are two basic approaches to scientific computing on parallel machines: 1) high-level tools and compilers that hide the message passing details making it easy to use but also tending to hide the pitfalls that lead to bottlenecks; 2) low-level tools for message passing that create scaleable code but require knowledge of algorithms and software that make them difficult to use. The frame-based toolkit approach is fundamentally different from both of these. It provides a parallel algorithm "template" upon which the physics of the problem can fill out stubs on the template. These take the form of callbacks or objects and serve to particularize the template to suit the application.

A PDF reacting flow application has been implemented in this frame-based method and the results look promising. The calculation is for a freely expanding H2/Air jet flame. The detailed chemistry consists of 9 species and accounts for 22 days of execution time on a state-of-the-art workstation, while the second-order-closure fluid mechanics model accounts for 15 minutes of this time. With the toolkit running on 10 workstations, the execution time is reduced to 3 days total. Predicted values of NOx emission agree very well with experiment and show that detailed chemistry is necessary to understand NOx formation in turbulent flow.

***Chen, J.-Y. (University of California, Berkeley):** "Development of Reduced Chemistry for CFD Applications"

Simplified chemical reaction mechanisms are often required for simulation of reacting flows to reduce computational time. Recent development of simplified mechanisms for complex combustion processes is based on systematic reduction of the detailed chemical mechanism rather than curve fits to limited experimental observations. Consequently, these newly developed features are capable of capturing many salient features of the detailed chemical kinetics. More importantly, for CFD applications, these features of the detailed mechanism can be obtained by a small number of scalars. This presentation will be given from the point view of CFD developers and users. The concept of systematic reduction method will be introduced with a simple thermal *NO* reaction mechanism. A step by step reduction of the detailed mechanism will be illustrated for hydrogen and methane air combustion. Automation of the procedures for constructing reduced reaction mechanisms by a computer program is currently being developed. Results for hydrogen combustion with *NOx* formation will be presented.

***Chima, R. V. (NASA Lewis); Giel, P. W. (NASA Lewis); and Boyle, R. J. (NASA Lewis):** "An Algebraic Turbulence Model for Three-Dimensional Viscous Flows"

An algebraic turbulence model is proposed for use with 3D Navier-Stokes analyses. It incorporates features of both the Baldwin-Lomax and Cebeci-Smith models. The Baldwin-Lomax model uses the maximum of function f(y) to determine length and velocity scales. An analysis of Baldwin-Lomax model shows that f(y) can have spurious maximum close to the wall, causing numerical problems and non-physical results. The proposed model uses the integral relations to determine $\delta^* u_e$ and δ used in the Cebeci-Smith model. It eliminates a constant in the Baldwin-Lomax model and determines the two remaining constants by comparison to Cebeci-Smith formulation. Pressure gradient effects, a new wake model, and the implementation of these features in a 3D Navier-Stokes code are also described. Results are shown for a flat plate boundary layer, an annular cascade, and endwall heat transfer in a linear turbine cascade. The heat transfer results agree well with experimental data which shows large variations in endwall Stanton number contours with Reynolds number.

****Chitsomboon, Tawit (ICOMP)**: "Computation of HSCT Ejector Flow Field"

The mixer-ejector has been a major candidate for noise suppression in the exhaust jet of the proposed High Speed Civil Transport (HSCT) due to its capability to slow down the jet velocity through fast mixing of the jet and the ambient air. Two computer codes, MAWLUS and PARC, have been used to simulate the flow field of this 3D, geometrically complex, turbulent, shear-driven flow. Some results will be shown together with certain discussions on basic CFC (about stability and convergence rate of implicit schemes).

Dang, Thong Q. (Syracuse University): "Inverse Method for Turbomachine Blades by the Circulation Method"

A recently developed fully three-dimensional inverse method for turbomachine blades, called the Circulation Method, is proposed for use in the design of rocket engine turbomachine components. In this inverse method, the primary prescribed flow quantity is the pitch-averaged tangential velocity component in the bladed region (i.e. the blade loading), and the primary calculated geometrical quantity is the three-dimensional blade profile.

In this presentation, we will demonstrate the extension of the basic 3D theory for infinitely-thin blades and inviscid flows to 1) handle blades with finite thickness, and 2) include viscous effects via the body-force model of Denton. Examples include the design of inlet guide vanes, compressor blades, and the Rocketdyne high-flow coefficient inducer.

***DeAnna, Russell G. (Army)**: "Direct Numerical Simulation of Boundary Layer Flow Over Surface Roughness"

Results from a *direct numerical simulation* of transitional flow over a surface with spherical roughness elements of height κ and a surface with random roughness of maximum height $\pm\kappa_{max}$ are presented. Periodic boundary conditions in the streamwise and spanwise directions simulate an infinite array of roughness elements, while a *body force*, designed to yield the streamwise Blasius velocity in the absence of roughness, maintains the flow. At $\kappa/\delta^*=0.72$, the mean velocity field in the spherical roughness domain contains secondary flow patterns within the region below 2κ, for Reynolds numbers, $U_\kappa\kappa/\upsilon$, between 90 and 225. The streamwise vorticity at these low Reynolds numbers is simply a result of the fluid's continuity and does not indicate rotating fluid or effects of inertia. The spheres distort the original Blasius profile into a mildly inflected layer containing low-momentum regions behind each sphere. These regions engender unsteady disturbances near the wall; however, the distribution of body force with vertical position above the wall is such that growth is suppressed in this region. Growth does occur in the unstable layer above the spheres where the body force is larger. The disturbance frequency is fixed by both the mean, streamwise velocity in the most unstable layer and the spacing between spheres; it is not the blunt-body vortex-shedding frequency expected for isolated bodies. When an oscillating component was added to the steady, Blasius body force, the response was independent of both forcing frequency and amplitude and, once again, depended on the mean velocity and the spacing between spheres.

****DeBonis, J. R. (NASA Lewis):** "Analysis of a 2D Mixer/Ejector Nozzle for Noise Suppression of a Supersonic Transport"

A Computational Fluid Dynamics (CFD) study was performed to analyze the NASA/GE 2DCD Mixer/Ejector Nozzle. This nozzle is being designed for use with the proposed High Speed Civil Transport (HSCT). The nozzle is intended to suppress noise during takeoff of the HSCT. The PARC3D full Navier-Stokes solver was used with both an algebraic and a κ-ε turbulence model. Data presented include pressure and velocity distributions and mixing effectiveness for the baseline nozzle. The baseline nozzle showed a large overexpansion and a shock occurs in the mixing section which negatively affects the nozzle performance. Mixing between primary and secondary streams was incomplete. A parametric study to determine the optimum area ratio of the mixing section for thrust performance was completed. This study showed that a slightly convergent mixing section is desirable. Finally, an analysis of an alternate mixer configuration to improve the mixing will be presented.

Fujii, Kozo and Tamura, Yoshiaki (Institute of Space and Astronautical Science) and Ogawa, Takanobu, (Shimizu Company): "Unsteady Flow Simulations Using a FSA Zonal Method With a Moving Grid System"

Problems and requirements for a numerical scheme with a moving grid system are discussed, especially for conservation in both time and space dimensions. Then some practical applications of unsteady flow simulations (blast waves, train moving into a tunnel, etc) using the FSA (fortified solution algorithm) zonal method with a moving grid system are presented.

****Georgiadis, Nick (NASA Lewis):** "Full Navier-Stokes Analyses of Nozzles for Supersonic Transport Application"

Controlling the noise produced by the nozzles of supersonic transport engines is one of the major technological challenges facing the High-Speed Research (HSR) program. Computational Fluid Dynamics (CFD) codes have become a tool used extensively by researchers to analyze flowfields of promising low-noise nozzle concepts. However, before CFD can be relied upon to predict the details of complex nozzle flows, validation studies need to be conducted to determine the capabilities and limitations of currently available codes.

This presentation will discuss analyses of two reference nozzles using the PARC full Navier-Stokes code. The first is an axisymmetric plug nozzle that is to be tested in NASA Langley's Jet Noise Laboratory. The second is an ejector nozzle that was designed and tested for STOVL application, but is similar to the mixer-ejector nozzles currently being considered in the HSR program. The effects of turbulence model selection and grid resolution on flow solutions will be emphasized in the presentation.

***Giel, P.W. (Sverdrup); Sirbaugh, J. R. (Sverdrup); Lopez, I. (Army); and Van Fossen, J. (NASA Lewis):** "Three-Dimensional Navier-Stokes Analysis and Redesign of an Imbedded Bellmouth Nozzle in a Turbine Cascade Inlet Section"

A computational analysis of an imbedded bellmouth inlet was performed with the PARC code to identify and eliminate the source of measured pitchwise flow non-uniformity in the NASA LeRC Transonic Turbine Blade Cascade. The computational domain extended from the beginning of a constant span section to a plane just upstream of the cascade of turbine blades. Spanwise symmetry allowed modeling of just half of the span. The blockage and acceleration effects of the blades were accounted for by specifying a periodic static pressure exit condition interpolated from an RVC3D code isolated blade calculation. Calculations of the original geometry showed total pressure loss regions consistent in strength and in location to experimental measurements. An examination of the results shows that the distortions are caused by a pair of vortices that originate as a result of the interaction of the flow with the imbedded bellmouth. Computations were performed for an inlet geometry which eliminated the imbedded bellmouth by bridging the region between it and the upstream wall. This analysis indicated that eliminating the imbedded bellmouth eliminates the troublesome pair of vortices, resulting in a flow with much greater pitchwise uniformity.

***Goldman, Louis J. (NASA Lewis):** "Experimental Measurement of Reynolds Stress in a Turbine Stator Wake Using Laser Anemometry"

Laser anemometry (LA) is a well established technique for the nonintrusive measurements of the mean velocity field within turbomachinery components. As an added bonus, turbulence information can also be extracted from these velocity measurements even for a single component LA system. About ten years ago, I described how to do this for two-dimensional flow using parameter estimation techniques. This not only allowed the turbulence quantities to be determined but also provided estimates of their uncertainties.

In the present talk, I will generalize this method to three-dimensional flow and thus to the determination of the complete Reynolds stress tensor. Prediction analysis has been used to estimate the uncertainties in the measurement of the Reynolds stress terms and these results will be presented. Some preliminary experimental measurements will also be discussed.

****Goodrich, John W. (NASA Lewis):** "Higher Order Finite Difference Methods for CAA"

Acoustic problems have become extremely important in recent years because of the High Speed Civil Transport research effort. The demands within CAA for extreme accuracy over relatively long times and distances require the development of new algorithms for useful simulation of aeroacoustic problems. This talk will present and compare four new high order finite difference methods applied to a model acoustic problem in order to demonstrate that finite difference techniques can be developed for productive application to CAA problems. The model problem is acoustic propagation with the Linearized Euler Equations in one space dimension for mean flows of Mach 0 and Mach 2. The four new algorithms are all usable for both subsonic and supersonic mean flows without modification. The algorithms are all local single step explicit finite difference methods, with the same order of accuracy in both space and time, with fixed central stencils, and with no parameters except the mesh size, Mach number, and raio of time to space mesh sizes. The four algorithms are of fourth, fifth, sixth, and seventh order accuracy respectively. The fifth and seventh order methods have diffusive leading order tuncation error terms, while the fourth and sixth order methods are dispersive. These algorithms will be exercised by acoustic propagation to nondimensional times of 5, 50, 500, and 5000. These algorithms all have isotropic extensions to two or three space dimensions, with no additional error or distortion due to grid orientation. Sample results for the scalar wave equation in two space dimensions will be presented.

Gunzburger, Max D. (ICOMP): "Analysis of Least-Squares Finite Element Methods for the Navier-Stokes Equations"

Least-squares finite element methods for incompressible viscous flows have been receiving considerable attention. Much success has been achieved in the algorithmic development and implementation of these methods. We discuss the results of some recently completed analyses for these methods. In particular, we focus on velocity boundary conditions in which case it is shown that in order to obtain optimally accurate solutions, one must introduce mesh dependent weights into the least-squares functional. The necessity of the use of weights is also demonstrated with computational experiments.

****Hagstrom, Thomas (ICOMP)**: "Numerical Radiation Conditions and Unsteady Flow Problems"

A typical feature of physical problems involving wave propagation is the radiation of energy to the "far field". The numerical solution of such problems requires the introduction of an artificial boundary. The boundary condition at this boundary must simulate the physical radiation effects. The design of boundary conditions for the standard equations of mathematical physics (wave equation, Euler equations, Navier-Stokes equations) has a long history and a body of practical experience exists. Surprisingly, though, the error analysis of these approximations is poorly developed. In this talk we outline a theory of approximate boundary conditions and consider its application to the standard problems. In the process we develop some new conditions as well as some surprising (and, we hope, illustrative) error estimates. We also present work in progress on high-order conditions for the Euler equations and on the compressible Navier-Stokes equations at low Mach number.

****Hardin, J. C. (NASA Langley):** "An Acoustic/Viscous Splitting Technique for Computational Aeroacoustics"

A new technique for the numerical analysis of aerodynamic noise generation is developed. The approach involves first solving for the time-dependent incompressible flow for the given geometry. A "hydrodynamic" density correction to the constant incompressible density is then calculated from a knowledge of the incompressible pressure fluctuations. The compressible flow solution is finally obtained by considering perturbations about the "corrected" incompressible flow. This fully nonlinear technique, which is tailored to extract the relevant acoustic fluctuations, appears to be an efficient approach to the numerical analysis of aerodynamic noise generation, particularly in viscous flows. Applications of this technique to some classical acoustic problems of interest, including some with moderately high subsonic Mach numbers, are presented to validate the approach. The technique is then applied to a fully viscous problem where sound is generated by the flow dynamics.

****Hariharan, S. I. (ICOMP)**: "On the Time Accurate Calculations of Exterior Aerodynamics and Aeroacoustics Problems"

This talk addresses 2D exterior problems governed by the Euler equations. The goal is to treat unsteady flows numerically. Difficulties arise in 2D problems which are computationally less intensive than their corresponding 3D counter parts. These difficulties are attributable to the decay properties of the associated wave equations. For example, linearized Euler equations governed by a wave equation with a convective derivative along the direction of the flow. We review a model governed by the 2D wave equation and present key ideas of stability of the solutions over a long time. These ideas then suggest a procedure for the construction of boundary operators on artificial boundaries that are designed to maintain the time accuracy of calculations. We present numerical examples to demonstrate the theory using a flow past an impulsively started cylinder at high subsonic Mach numbers.

****Hayder, M. Ehtesham (ICOMP)**: "Jet Flow Simulations for Near Field Sound Source Computations"

In this talk, numerical issues regarding a recently developed multidimensional jet flow code will be addressed. This numerical model is based on a fourth order extension of MacCormack's scheme. Improvement has been made to the original extension developed by Gottlieb and Turkel. Outflow boundary conditions are very important for jet flow calculations. A summary of numerical experiments with a few boundary treatments will be presented. Preliminary 3D results will also be presented.

***Huang, P. G. (Eloret Institute)**: "Turbulence Modeling for Compressible Flows, Part I. — Modeling Mixing and Boundary Layer Flows"

The present seminar centers on the "dissipation-transport" equation and its role in predicting the compressible law of the wall. First, a skin friction and velocity profile family for compressible turbulent boundary layers is developed. The profile family has been compared with a range of high speed flow data with great success, including supersonic and hypersonic experiments and a recent compressible channel flow DNS.

Predictions of the velocity profiles using standard turbulence models have shown that the unmodified models have given rise to too small a value of the von Karman constant, κ, in the log-law region. Thus, if the models are otherwise accurate, the "wake" component is over-predicted and the predicted skin friction is lower than the expected value. The magnitude of the errors that results from neglecting the dependence on density depends on the variables used to specify the length scale.

To agree with experimental values for κ in the compressible boundary layer, the apparent eddy viscosity must be increased. This "compressible effect" - which is an artifact of conventional turbulence modeling rather than something real - is exactly opposite to that in mixing layers, where the growth rate, and by implication the eddy viscosity, decreases with increasing Mach number. As a consequence, recent compressibility modifications proposed for the mixing layer have increased the errors in predictions of flat plate compressible boundary layer flows.

***Huang, P. G. (Eloret Institute)**: "Turbulence Modeling for Compressible Flows, Part II. — Complex Flow Computations"

Calculations of high Mach number turbulent flows have become a major challenge in CFD in recent years. In the present seminar, attention will be focused on some of the recent activities at NASA Ames on predictions of flows with complex shock-wave/boundary-layer interactions. Numerical methods to solve the mean flow and turbulence equations, including Reynolds stress transport models, will be discussed. Comparisons of low-Reynolds-number and wall-function techniques will also be made.

The seminar presents results of calculations for a range of 2D and 3D compressible turbulent flows using both two-equation and Reynolds stress transport models. Comparisons with the experimental data have shown that baseline models underpredict the extent of flow separation but over-predict the heat transfer near flow reattachment. Modifications to the models are described which remove the above-mentioned deficiencies.

***Hsu, Andrew (Sverdrup)**: "A PDF Approach for Compressible Turbulent Reacting Flows"

The objective of the present work is to develop a probability density function (pdf) turbulence model for compressible reacting flows for use with a CFD flow solver. The probability density function of the species mass fraction and enthalpy are obtained by solving a pdf evolution equation using a Monte-Carlo scheme. The pdf solution procedure is coupled with a compressible CFD flow solver which provides the velocity and pressure fields. A modeled pdf equation for compressible flows, capable of capturing shock waves and suitable to the present coupling scheme, is proposed and tested. Convergence of the combined finite-volume Monte-Carlo solution procedure is discussed, and an averaging procedure is developed to provide smooth Monte-Carlo solutions to ensure convergence. Two supersonic diffusion flames are studied using the proposed pdf model and the results are compared with experimental data; marked improvements over CFD solutions without pdf are observed. Preliminary applications of pdf to 3D flows are also reported.

***Johansson, Arne V. (Royal Institute of Technology, Stockholm)**: "Modeling of Intercomponent Transfer in Reynolds Stress Closures of Homogeneous Turbulence"

In *classical* Reynolds stress closures of turbulent flows, transport equations are formulated for the Reynolds stress tensor and dissipation rate. For homogeneous turbulence, there is no spatial redistribution of energy and the modeling difficulties lie in the treatment of ε-equation and the intercomponent transfer processes. The transport equations for $\overline{u_i u_j}$ may be replaced by equations for the kinetic energy, κ, and stress anisotropy tensor a_{ij} ($\equiv \overline{u_i u_j}/\kappa - 2\delta_{ij}/3$) to effectively separate the "amplitude" related issues involved in the prediction of κ and ε from the relative distribution among the components described by the normalized, traceless tensor a_{ij}.

Intercomponent transfer in the transport equations for a_{ij} (or $\overline{u_i u_j}$) is represented by $\Pi_{ij}^{(r)}$, $\Pi_{ij}^{(s)}$ denoting the so called rapid and slow pressure strain rate terms, respectively, and a term related to the effects of anisotropic dissipation rate. Detailed data for the individual terms obtained from direct numerical simulations and physical experiments are presented and used in the evaluation of the modeling ideas and other aspects of intercomponent transfer.

New experimental results will be presented where a pair of specially built hot-wire X-probes have been used to investigate the anisotropy of the dissipation rate in axisymmetric turbulence. The trends are well described by the model proposed by Hallback *et al.* (Phys. Fluids (1990) both for the simulation results and for the higher Reynolds number data obtained from the physical experiments. For low turbulence, Reynolds numbers the simulations have shown the slow part of the pressure-strain rate to be strongly suppressed. A simple model for the variation of Rotta constant has been shown to capture this Reynolds number variation. Recently developed modeling ideas for the rapid pressure-strain rate will also be discussed. Strong realizability and kinematic constraints have been used to derive tensorially correct forms of the model, and the model constants have been calibrated by use of rapid distortion theory.

***Koshioka, Yasuhiro (Fuji Heavy Industries, Japan):** "A Numerical Simulation of Two-Dimensional Transient Shear Layer"

A numerical algorithm was developed based on Arakawa's algorithm, that is constructed along the idea of the integral constraints on quadratic quantities of inviscid, incompressible two-dimensional flow, such as conservation of mean kinetic energy and mean square vorticity (enstrophy). The importance of this idea is shown through the numerical simulation of the two-dimensional shear layer. In this simulation, the development of small disturbances in the shear layer and the merging process of vortices is clearly observed. Finally, the extension of this algorithm to the three-dimensional flow is shown and the simulation of two-dimensional flow that contains three-dimensional disturbance is carried out by this algorithm.

***Lakshminarayana, Budugur (Pennsylvania State University); and Fan, Sixin (Pennsylvania State University):** "Turbulence Modeling and Computation of Unsteady Flows due to Rotor/Stator Interaction in Axial Turbomachines"

The unsteady pressure field and the boundary layers on a turbomachinery blade row resulting from the periodic wake-passing from the upstream blade rows are investigated. A time accurate Euler solver has been developed using an explicit four-stage Runge-Kutta scheme. Two dimensional unsteady non-reflecting boundary conditions are used at the inlet and the outlet of the computational domain. The unsteady Euler solver captures the wake propagation and the resulting unsteady pressure field, which is then used as the input for a 2-D unsteady boundary layer procedure to predict the unsteady response of blade boundary layers. The boundary layer code includes an advanced κ-ε model developed for unsteady turbulent boundary layers. The present computational procedure has been validated against analytic solutions and experimental measurements.

Numerical simulations are carried out to understand the effects of wake velocity defect and turbulence intensity on the development of the blade boundary layers, including the unsteady transition process. Large unsteadiness in the local skin friction coefficient arises from the time dependent transition process in the boundary layer, which is mainly induced by the high turbulence intensity in the incoming wake. A parametric study is then carried out to determine the influence of design parameters on the development of unsteady blade boundary layers, such as the axial gap between rotor and stator, the rotor/stator blade ratio and the wake inflow angle. It is shown that the unsteadiness in the blade boundary layer increases with a decrease in the axial gap, an increase in rotor/stator blade ratio or an increase in wake inflow angle. The time averaged boundary layer momentum thickness at the trailing edge of the blade is found to increase significantly over the steady value for higher rotor/stator blade ratio and the larger wake inflow angle. Increase of the rotor/stator blade ratio results in higher frictional drag of the blade.

Leonard, B. P. (ICOMP): "Some Things (About CFD) They Didn't Teach You in School -- and Some Others They Did That Were Wrong!"

As you were probably taught in school, ("standard") second-order central differencing is wiggly or unstable for CFD (right!); hence:

- There is a "need" for artificial viscosity or diffusivity (equivalent to first-order locally). Wrong! Leading truncation error should be dissipative, yes, *but not diffusive* (so as not to interfere with modelled physical diffusion); to lowest order:
- Third-order methods (QUICK and QUICKEST) are the appropriate basis for CFD.

But you may have learnt that:

- "Higher-order methods are too expensive; corresponding BC's are too difficult." This is (to put it politely) bull-dust; in fact:
- Computational efficiency *increases monotonically* with order.
- Higher-order numerical BC's are no harder than low-order.

You learnt about "central" and "upwind" schemes. It turns out that:

- "Central" schemes are actually downwind weighted; most "upwind" schemes use cell-centered *symmetric* interpolation.

Everyone "knows" that:

- "The CFL condition is a stability constraint on the time step." Actually, it's not. As a matter of fact:
- The CFL condition is merely a range restriction on interpolants.
- There is *no time-step stability constraint* for explicit convection schemes.

They probably didn't teach you:

- The difference in truncation error between single-point and finite-volume formulations; e.g., CDS diffusion is *twice* as accurate in FV as it is in SP (the exact same discrete operator).

Finally (since it's only just been done) you wouldn't have heard about:

- Recent progress in constructing *genuinely multidimensional* limiters for highly advective flow.

***Liou, William (ICOMP)**: "Weakly Nonlinear Models for Turbulent Free Shear Flows - Linear Instability of Curved Free Shear Layers"

Turbulence closure schemes based on a weakly nonlinear theory with a description of the dominant large-scale structures as instability waves have been applied successfully in the prediction of various plane and axisymmetric free shear flows. In order to extend the wave model to curved mixing layers, a linear stability analysis has been performed. Two mean velocity profiles that represent stably and unstably curved free mixing layers were considered. In this presentation, results of the linear instability study for five curvature Richardson numbers are described. The instability characteristics of the mixing layer were found to vary significantly with the introduction of the curvature effects. The results also indicate that, in a manner similar to the Görtler vortices observed in a boundary layer along a concave wall, instability modes of spatially developing streamwise vortices pairs may appear in an unstably curved mixing layer.

***Liu, Chaoqun (University of Colorado, Denver):** "Direct Numerical Simulation of Flow Transition in 3-D Boundary Layers."

Flow transition is one of the fundamental, unsolved problems in modern fluid mechanics. The existing numerical studies are quite limited:

1. Most of them use a temporal approach which is not physical,
2. The codes blow up at the flow breakdown stage before transition (pre-onset simulation only),
3. One single 2-D bump in 2-D boundary layer to simulate surface roughness,
4. Very expensive in cpu cost (around 100 - 1000 YMP hours for a 3-D flat plate).

Liu's work is quite different:

1. Spatial approach (fully implicit, multigrid, line-distributive relaxation, high-order finite difference, and new buffer outflow boundary treatment),
2. Able to simulate the whole process of transition including linear evolution, nonlinear instability, breakdown and transition. It provides detailed and visualized description of flow breakdown and transition,
3. Able to simulate 3-D boundary layers with multiple 3-D roughness elements (random shape and random distribution in curvilinear coordinates)
4. Acceptable cpu cost (in the range of 1-6 YMP hours).

Liu's results show good agreement with linear stability theory, secondary instability theory, and a number of experiments.

****Mankbadi, Reda R. (NASA Lewis):** "Computational Aero-Acoustics (CAA): Large-Eddy Simulations of a Supersonic Jet and Its Radiated Sound"

Computational Aero-Acoustics (CAA) is concerned with calculations of the aerodynamically-generated sound source and sound propagation. The flow and sound fields are calculated starting from the time-dependent governing differential equations. This involves large-eddy simulations to capture the time-dependent sound source, algorithm development and evaluation for CAA, boundary conditions appropriate for unsteady-state solution, and differential/integral techniques to study the propagation of sound.

Results will be presented for large-eddy simulation of a supersonic jet with emphasis on capturing the unsteady features of the flow pertinent to noise emission. A high-order accurate numerical scheme is used to solve the filtered equations. For random in-flow disturbances, the wave-like feature of the large-scale structure is demonstrated.

Several approaches to obtain the far-field sound from the near field will be discussed. Namely:

(a) Direct extension of the nonlinear-solution domain to the far field
(b) Acoustic analogy
(c) Kirchhoff
(d) Linear Euler equation

***Matsuo, Yuichi (National Aerospace Laboratory):** "Computations of Separated Aerodynamic Flows Using a Modified Near Wall κ-ε Model"

Complex aerodynamic flows with separation were computed by using a near wall κ-ε model. The model is modified so that the near wall damping functions do not include friction velocity and so that the predicted turbulence quantities are matched to the variations from DNS data. In order to apply the model to practical flows, an efficient and robust flow solver was developed. The code is based on a finite difference approach where a state-of-the-art upwind scheme is used for the convection terms and further source terms of the turbulence quantities are treated implicitly. Some representative computations were carried out for flows over backward facing step, a NACA0012 airfoil, a supersonic compression corner and a ONERA M6 wing. The last one is a 3D case. The results show that the present model provides satisfactory predictability even for the complex flows with separation.

***Norris, A. T. (Cornell University)**: "The Application of PDF Methods to the Modeling of Turbulent Diffusion Flames"

The model transport equation for the joint probability density function (jpdf) of velocity and dissipation provides a closed set of equations for the modeling of turbulent flows. Features of this method include the presence of length and time scales, intermittency and the exact treatment of convection. This approach is extended to reactive flows by including composition, (i.e. solving for the jpdf of velocity, dissipation and composition) which has the feature that reaction is treated exactly.

A particle-based Monte Carlo scheme is used to solve the transport equation, for several different classes of flow of increasing complexity. The simplest flow considered is a variable density plane mixing layer while the most complex is a piloted CO/H2/N2 - air diffusion flame close to extinction. Details of the modeling, including the molecular mixing models and reduced thermochemical mechanisms, are discussed for each applicable flow.

****Panda, J. (NASA Lewis, Resident Research Associate)**: "Measurement of Shock Oscillation in Underexpanded Jets Using a New Optical Technique"

The oscillations of various shock surfaces in underexpanded free jets of fully expanded Mach numbers in the range of 1.2 to 1.8 are studied through an optical technique. The technique is based on the new observation that, when a laser beam is passed tangential to a shock wave, a portion of the beam spreads out in a diffraction like pattern normal to the shock surface. The physical reason behind this optical phenomenon is, so far, not well understood.

By scanning a shock containing plume with a Laser beam and sensing the diffracted light by a photomultiplier tube, the locations and the amplitudes of oscillations of various shock surfaces are measured. The time averaged shock positions, identified by this optical technique, agree well with those identified by the Schlieren photographs and LDV measurements. The amplitudes of oscillation of all shock cells except for the first one or two are found to be unexpectedly large. As a shock oscillates, the diffracted light appears or disappears. By measuring the time variation of this light, it is observed that the entire surfaces of all shocks oscillate primarily at the screech frequency. Suppression of screech by a reflector plate upstream of the nozzle exit is found to diminish the periodic shock motion but the background, random component of oscillation persists. Measurements of the phase of shock oscillation, obtained by cross-correlation with the jet screech, show that the first shock cell in the M_j = 1.4 jet oscillates in a spinning mode. The probable reason of shock oscillations due to the pressure variations associated with the large scale organized structures will be discussed.

Pletcher, Richard H. (ICOMP): "Unsteady Viscous Flows: Computational Strategy and Some Recent Results"

Studies are underway at Iowa State University to develop accurate and efficient numerical modeling for a number of unsteady viscous flows. A strategy known as preconditioning is being employed to efficiently compute flows over a wide range of Mach numbers. The what, why, and how of preconditioning will be discussed briefly from a physical point of view, and the properties of such schemes as applied to two unsteady flows, namely the impulsively started driven cavity and the startup of pipe flow will be described. Preliminary results of an extension of this all-speed approach to the large eddy simulation (LES) of turbulent flows will be presented. As time permits, early results of another extension to treat free surface flows in the form of "surface capturing" will also be presented.

****Roe, Philip (University of Michigan):** "Algorithms for Long-Range Wave Propagation"

There are many problems, of a size demanding the power of supercomputers, that involve the propagation of small-amplitude waves over distances that are large compared with the wavelength, but not so large that the simplifications of geometrical optics can be applied. Examples include aeroacoustics, electro-magnetic waves, medical imaging, and material testing. Substantial effort is being put into developing CFD-type methods for such problems. The emphasis in these algorithms is not on the correct handling of nonlinearities and discontinuities, but on the accurate resolution of high frequency waves by the minimum number of grid points. Some of the current approaches will be surveyed, such as compact, spectral, Hermitian and discontinuous Galerkin schemes.

Then a method under development by the speaker will be described. Its key elements are an advection scheme that is compact, upwinded, and time reversible, so that no dissipation occurs, and a corresponding discretization of the bicharacteristic equations on a staggered mesh. The method will be applied to a test problem in linear acoustics involving a speaker oscillating within an infinite baffle. Excellent results using either seven or four points per wavelength are obtained from the second- and fourth-order versions respectively.

Finally some remarks will be offered on the discrete boundary conditions that are required for such calculations.

***Shih, Tsan-Hsing (ICOMP) and Hsu, Andrew T. (Sverdrup)**: "Effect of the Coriolis Force on Compressible Turbulence"

Direct numerical simulation results and theoretical analysis are presented for the effect of the Coriolis force on compressible homogeneous isotropic turbulence. It is shown that the Coriolis force serves as a frequency modulator on turbulence. While the Coriolis force neither creates nor destroys turbulent kinetic energy, it redistributes energy by eliminating low frequency waves and transferring energy to waves with a frequency of 2Ω. The dissipation rate of turbulent kinetic energy can be either reduced or enhanced depending on whether or not the ratio between the rotation time scale and the Kolmogorov time scale is much greater than one. It has been demonstrated both theoretically and numerically that the Taylor-Proudman theorem is applicable to homogeneous turbulence only when the time scale of rotation, defined as the inverse of the frequency of the inertial waves, approaches the Kolmogorov time scale, and that two-dimensionalization occurs in this regime.

***Stewart, Mark (Sverdrup)**: "Towards the Simulation of a Full Turbofan Engine in the Meridional Plane"

The numerical simulation of the aerodynamics of a full jet engine is a problem of interest in engineering research and design. Existing analysis techniques deal with individual components and largely neglect intercomponent effects. Yet the aerodynamic performance of a jet engine depends on these components working together efficiently.

In this simulation of a jet engine, the 3D flow equations are averaged to axisymmetric flow equations, defined in the 2D meridional plane of an engine. The engine's meridional plane is covered with a multiblock grid which resolves blades and other components. The meridional plane includes external flow outside The engine and the internal ducts. The turning effects of blades, combustion, blockage, losses and real gas effects are represented in the equations with terms and interior conditions.

Some numerical, modeling and physical issues in this simulation will be discussed including accounting for losses, imposing internal conditions and the numerical stability of large multistage compressors and turbine components.

The method will be demonstrated with two examples. Convergent numerical solutions will be shown for the 1.15 Pressure Ratio Fan Engine model. The applicability of these methods to commercial engines will be demonstrated with the Energy Efficient Engine.

****Stoker, Robert (Georgia Institute of Technology)**: "An Evaluation of Finite Volume Direct Simulation and Perturbation Methods in CAA Applications"

A number of questions have emerged in the field of Computational Aeroacoustics (CAA) over the capability of applying conventional Computational Fluid Dynamics (CFD) methodologies to capture the physical attributes of acoustic fields. one of the more fundamental questions is the capability of direct simulation techniques versus perturbation techniques to capture the acoustic phenomena. In some work done at the Georgia Tech Research Institute these questions have been addressed by applying both a direct simulation and a perturbation method to some standard acoustic problems. The effects of grid spacing, dissipation, frequency and amplitude have been studied and compared for both methodologies.

Szeri, Andras Z. (University of Pittsburgh): "Flow Between Rotating Cylinders"

Flows induced by rotating coaxial cylinders of infinite length have been the subject of numerous investigations over the last few decades. We generalized the geometry of this flow recently to include cylinders of arbitrary eccentricity and finite length. In the numerical study I present here the relevant steady state governing equations are solved in a mixed formulation, employing Galerkin's method with B-spline discretization, that circumvents the Babushka-Brezzi stability condition. To study solution branching and multiplicity we employ parametric continuation techniques.

Motivation for the work comes from my studies in journal bearing lubrication. Journal bearings (eccentrically rotating coaxial cylinders) of large rotating machinery often operate at speeds inconsistent with laminar flow of the lubricant: turbulence models are employed under such circumstances to predict bearing performance. What makes this analysis particularly difficult is the lack of criteria for the loss of stability of the basic flow, except in the highly unrealistic case of zero load (concentric geometry) and isothermal operation.

Tryggvason, Gretar (ICOMP): "Direct Simulations of Multi-Phase Flows"

Multi-fluid and multi-phase flows are of importance in many engineering processes. Sprays and boiling heat transfer are classical examples. Direct computations of such flows have remained one of the frontiers of Computational Fluid Dynamics due to the difficulty in handling moving fluid interfaces. We have recently developed a numerical technique that holds considerable promise for these challenging problems. The technique is based on an explicit tracking of the interface and a relatively conventional finite difference approximation to the Navier-Stokes equations. The method is applicable to both two- and three-dimensional flows. Several results are presented, including simulations of collisions and coalescence of drops and the rise of many buoyant bubbles.

Tumin, A. (Tel-Aviv University, Israel): "Application of Biorthogonal Eigenfunction Systems to Hydrodynamic Stability Problems"

Biorthogonal eigenfunction systems for linearized Navier-Stokes equations and spatially growing disturbances are formulated. Two- and three-dimensional boundary layers in an incompressible fluid and in a compressible gas are considered. As examples of the biorthogonal eigenfunction systems application the following problems are discussed:

1. Decomposition of a complex signal data obtained in an experiment when a partial information is available. The results are illustrated by experimental data analysis for a wall-jet flow. Also a numerical example for cross-flow disturbances in a boundary layer over a swept-back wing is considered.

2. A perturbation method for analysis of a three-wave nonlinear interaction in a 3D compressible boundary layer is developed in terms of an averaging technique on the biorthogonal eigenfunction system basis. A numerical example for Tollmien-Schlichting wave interactions in a subsonic boundary layer over a swept-back wing shows an opportunity of an amplification for rather broad packets without an exact resonance synchronism.

3. Excitation of a normal mode by a localized forcing of boundary layer flow is briefly reviewed with the biorthogonal eigenfunction system help.

Viswanathan, K. (Penn State University): "Turbulent Mixing in Supersonic Jets"

This study examines the stability and mixing characteristics of high-speed jets, both in the presence and absence of confining walls. Strategies for enhancing mixing are explored. The unsteady motion of the shear layer is described using instability wave modeling. Two models are described for the turbulent mixing of compressible axisymmetric jets. The models assume that the mixing process is dominated by large scale coherent structures, which are modeled as a super-position of instability waves. The basis for the instability wave development is the integral form of the time-averaged kinetic energy flux equation. The current models emphasize the time-dependent motion of the shear layer, and a notable feature of these models is the absence of any model constants.

A highly supersonic unconfined jet possesses many modes of instability. The characteristics of these modes are examined and classified. The modifications of these instabilities by the presence of an external stream and an outer wall are studied in detail. The role of different instability modes in the jet dynamics is also examined. The instability wave modes of supersonic jets of non-circular cross sections confined in non-circular ducts are analyzed using the boundary element methods.

The results of this study indicate that the mixing and development of free shear layers is related closely to the spatial stability characteristics of the mean flow. The effects of compressibility are quite pronounced. The presence of outer walls are shown to have a strong effect on the instability modes of a high-speed jet. When the jet is confined in a circular duct, there is found to be a predicted increase in the initial spread rate of the jet.

Wu, Jie (University College of Swansea): "Automatic Generation of Unstructured Meshes in Arbitrary Domains"

A method is presented which generates unstructured triangular meshes based on the advancing front technique. The distinctive features of this method are that elements with large size variations can be generated and that local elongation of elements in certain prescribed direction can be achieved. The mesh generation is carried out in such a way that nodes and elements are created simultaneously. The desired mesh distribution is defined by piecewise linear interpolation on a background mesh formed by triangles. This makes the mesh generator particularly suitable for adaptive remeshing. With further modifications, a quadrilateral mesh generator is also developed.

***Yang, Zhigang (ICOMP)**: "On the Linear Stability of a Trailing Line Vortex"

The viscous linear stability of a trailing line vortex (Batchelor vortex) is studied. The flow is characterized by two parameters, the Reynolds number Re and the rotation rate q. The marginal stability curve which separates the stable domain from the unstable domain was searched over the (Re, q) plane. It is found that on the marginal stability curve, q increases with the Reynolds number and does not approach a constant even when the Reynolds number is as large as 10^5. The values of q for large Reynolds numbers are higher than the inviscid counterpart. These findings suggest that modes giving rise to the marginal stability are viscous and do not approach the inviscid limit as the Reynolds number goes to infinity. These modes have an azimuthal wavenumber $n = -1$ when the Reynolds number is large than 200, in contrast to $n = -2$ for small Reynolds numbers. As the Reynolds number is increased, the eigenfunctions of the marginal stability modes become more and more concentrated near the axis of the vortex, suggesting that these modes are viscous center modes in the limit of large Reynolds number.

****Zaman, Khairul (NASA Lewis):** "Experiments on Flow and Noise Fields of Axisymmetric Jets"

Past and recent experimental studies on the topic, in which the speaker was involved, are to be briefly discussed in two parts. Part I will be based on past experiments at LeRC on the flow-field as well as the near-and far-field sound of subsonic jets. This will include the effect of acoustic excitation on the jet noise. Part 2 will be based on recent experiments at LeRC on the influence of vortex generators on free jets. Starting with the effect of such a device on the screech noise from supersonic jets, attention will be focused mainly on the flow-fields. An issue to be addressed is the generation mechanism and the subsequent evolution of streamwise vorticity in the flow under consideration.

***Zhu, Jiang (ICOMP) and Shih, Tsan-Hsing (ICOMP):** "A Realizable Reynolds Stress Algebraic Equation Model for Complex Flows"

The invariance theory in continuum mechanics is applied to analyze Reynolds stresses in high Reynolds number turbulent flows. The analysis leads to a turbulent constitutive relation that relates the Reynolds stresses to the mean velocity gradients in a more general form in which the classical isotropic eddy viscosity model is just the linear approximation of the general form. On the basis of realizability analysis, a set of model coefficients is obtained which are functions of the time scale ratios of the turbulence to the mean strain rate and the mean rotation rate. These coefficients will ensure the positivity of each component of the turbulent kinetic energy - realizability that most existing turbulence models fail to satisfy. Separated flows over backward-facing step configurations are taken as applications. The calculations are performed with a conservative finite-volume method. Grid-independent and numerical diffusion free solutions are obtained by using differencing schemes of second-order accuracy on sufficiently fine grids. The calculated results are compared in detail with the experimental data for both mean and turbulent quantities. The comparison shows that the present proposal significantly improves the predictive capability of κ-ε based two equation models.

***Zhu, Jiang (ICOMP) and Shih, Tsan-Hsing (ICOMP):** "Computation of Confined Coflowing Jets with Three Turbulence Models"

A numerical study of confined jets in a cylindrical duct is carried out to examine the performance of two recently proposed turbulence models: an RNG-based κ-ε model and a realizable Reynolds stress algebraic equation model. The former is of the same form as the standard κ-ε model but has different model coefficients. The latter uses an explicit quadratic stress-strain relationship to model the turbulent stresses and is capable of ensuring the positivity of each turbulent normal stress. The flow considered involves recirculation with unfixed separation and reattachment points and severe adverse pressure gradients, thereby providing a valuable test of the predictive capability of the models for complex flows. Calculations are performed with a finite-volume procedure. Numerical credibility of the solutions is ensured by using second-order accurate differencing schemes and sufficiently fine grids. Calculations with the standard κ-ε model are also made for comparison. Detailed comparisons with experiments show that the realizable Reynolds stress algebraic equation model consistently works better than the standard κ-ε model in capturing the essential flow features, while the RNG-based κ-ε model does not seem to give improvements over the standard κ-ε model under the flow conditions considered.

1993 Lewis Internal Workshop on Computational Turbulence Modeling

September 15-16, 1993

1. Objectives

The purpose of this meeting is to discuss **the current status** and **future development** of turbulence modeling in computational fluid dynamics **for aerospace propulsion systems**. Various turbulence models have been developed and applied to different turbulent flows over the past several decades and it is becoming more and more urgent to assess their performance in various complex situations. In order to help users in selecting and implementing appropriate models in their engineering calculations, it is important to identify the capabilities as well as the deficiencies of these models. This also benefits turbulence modelers by permitting them to further improve upon the existing models.

This workshop is designed for exchanging ideas and enhancing collaboration between different groups in the Lewis community who are using turbulence models in propulsion related CFD. In this respect this workshop will help the Lewis goal of excelling in propulsion related research.

2. Format

- This meeting has seven sessions for presentations and one panel discussion over a period of 2 days.
- Each presentation session is assigned to one or two branches (or groups) to present their turbulence related research work. Each group should address at least the following points: current status of turbulence model applications and developments in the research; progress and existing problems; requests about turbulence modeling.
- Each speaker will be given 18 minutes for the presentation which will be followed by 2 minutes of questions/answers.
- The panel discussion session is designed for organizing committee members to answer management and technical questions from the audience and to make concluding remarks.
- All the talks will be collected and printed in a proceedings of this workshop.
- Workshop is open to all Lewis related personnel.

3. Organizing Committee

L.A. Povinelli (Chairman)	T. Keith	R.J. Shaw
J.M. Abbott	R. Mankbadi	R.J. Simoneau
P.G. Batterton	E.J. Mularz	R.M. Stubbs
R.E. Gaugler	D.R. Reddy	A. Shabbir
		T.-H. Shih

Workshop Agenda

Welcome by L. Povinelli

Turbulence Modeling Activities at CMOTT

Chairman: T.-H. Shih

CMOTT Research Activities by T.-H. Shih

Research on two equation models by Z. Yang

A Reynolds stress algebraic equation model by J. Zhu

Assessment and development of second order turbulence models by A. Shabbir

A multiple-scale model for compressible turbulent flows by W. Liou and B. Duncan (CFD branch)

PDF models by A. Hsu (CFD branch) and A. Norris

Analytical theories of turbulence applied to turbulence modeling by R. Rubinstein

DNS of turbulence, transition and effect of rotation by J. Van der Vegt and A. Hsu (CFD branch)

Heat Transfer and Turbomachinery Flow Physics Branches

Co-chairman: R.J. Simoneau and R. Gaugler

Introduction by R. Simoneau

Introduction by R. Gaugler

Modeling of by-pass transition by F. Simon

Algebraic models for turbine blade heat transfer by R. Boyle

Two-equation models for turbine blade heat transfer by A. Ameri

Thermal turbulence models for turbine blade heat transfer by J. Schwab

Aerothermochemistry and Computational Methods for Space Branches

Co-chairman: E. Mularz and R.M. Stubbs

Introduction: Development of New Flow Reactive Code (ALLSPD) by E. Mularz

A coupled implicit solution method for turbulent spray combustion in propulsion systems by K.-H. Chen and J.-S. Shuen

Research Activities by R.M. Stubbs

On turbulent transport of chemical species in compressible reacting flows and Unsteady transitional flows over forced oscillatory surfaces by S.-W. Kim

On the accuracy of compact differencing schemes for DNS by S.-T. Yu

Computational Fluid Dynamics Branch and Lewis Research Academy

Co-chairman: D.R. Reddy and R. Mankbadi

Introduction by D.R. Reddy

Turbulent back-facing step flow and the $k-\epsilon$ model: A critical comparison by C. Steffen

Analysis of supersonic flows using the $k-\epsilon$ models and the RPLUS code by J. Lee

Validation of a $k-\epsilon$ model in RPLUS2D code for non-reacting/reacting subsonic shear layers by H. Lai

Computational aeroacoustics as a branch of turbulence research by R. Mankbadi

Jet noise prediction using $k-\epsilon$ turbulence model by A. Khavaran

Numerical simulation of supersonic flow using $k-\epsilon$ model by S.H. Shih

Propulsion Systems Division

Chairman: P.G. Batterton

Introduction by P.G. Batterton

Subsonic inlet flows with transition by D. Hwang and K. Ahn

A comparative study of turbulence models in predicting hypersonic inlet flows by K. Kapoor

Turbomachinery flows by R. Chima

Propulsion Systems Division

Chairman: P.G. Batterton

Low emission combustors by J. Deur

Development of a reliable algebraic turbulence model giving engineering accuracy at reasonable cost by B.P. Leonard and J.E. Drummond (University of Akron)

Application of algebraic and two-equation turbulence models to HSR nozzle flow calculations by J. DeBonis and N. Georgiadis

Aircraft icing by M. Potapczuk

Applied RNG turbulence model for 3-D turbomachinery flows by K. Kirtley

Applied $k-\epsilon$ and Baldwin-Lomax turbulence models for S-ducts by G. Harloff

Inlet, Duct, and Nozzle Flow Physics Branch

Chairman: J. M. Abbott

Introduction/Overview by J. Scott

PROTEUS experience with the modified MML turbulence model by J. Conley

Turbulence model experiences for a round-to-rectangular transition duct by C. Towne

PROTEUS experience with three different turbulence models by T. Bui

Several examples where turbulence models fail in inlet flow field analysis by B. H. Anderson

Panel Discussion and Concluding Remarks

TABLE I. The ICOMP Research Staff - 1993

A. Resident Staff.

Kumud Ajmani, Ph.D., Mechanical Engineering, Virginia Polytechnic Institute and State University, 1991. Development of Codes for Parallel Processing. January, 1992--Present.

Tawit Chitsomboon, Ph.D., Mechanical Engineering, Old Dominion University, 1986. Code Development for Mixer-Ejector Nozzle Flows. January, 1990--Present.

Joongkee Chung, Ph.D., Mechanical Engineering, University of California, Berkeley, 1991. Code Development for Unsteady Inlet Flows Using Parallel Processing. May, 1992--Present.

Dominic A. R. Davis,. Ph.D., Fluid Mechanics, University College, London, 1991. Nonlinear Spatial Evolution of Instability Waves in Two-dimensional Free Shear Layers. October, 1991--August, 1993.

Robert J. Deissler, Ph.D., Physics, University of California, Santa Cruz, 1986. 1) Convectively Unstable Taylor-Couette Flow with a Noise-Sustained Spatially-Growing Wave Structure and 2) Instability During Marangoni Convection in a Fluid with Two Free Surfaces. November, 1990--September, 1993.

Bernard Greenspan, Ph.D., Applied Mathematics, Cornell University, 1981. New Solution Methods for PDE's Using Conservation Laws in Both Time and Space. December, 1992-- Present.

Ehtesham Hayder, Ph.D., Mechanical and Aerospace Engineering, Princeton University, 1988. Computation of Jet Noise in the Source Region Near the Nozzle Exit. February, 1991--Present.

Duane R. Hixon, Ph.D., Aerospace Engineering, Georgia Institute of Technology, 1993. Numerical Calculation of Jet Noise from First Principles. October, 1993--Present.

Lin-Jun Hou, Ph.D., Engineering Science and Mechanics. Georgia Institute of Technology, 1992. Flow Applications of the Least-Squares Finite Element Method. July, 1992--Present.

Bo-nan Jiang, Ph.D., Engineering Mechanics, University of Texas, Austin, 1986. Team Leader, Flow Applications of the Least-Squares Finite Element Method. October, 1987--Present.

Kai-Hsiung Kao, Ph.D., Aerospace Engineering Sciences, University of Colorado, 1989. Chimera Overset Grid Scheme with Time-Accurate 3D Compressible Finite Volume Navier-Stokes Flow Solver. November, 1992--Present.

William W. Liou, Ph.D., Aerospace Engineering, Pennsylvania State University, 1990. Modeling and Calculation of Compressible Turbulent Flows. November, 1990--Present.

James M. Loellbach, Ph.D. expected 1994, Aeronautical and Astronautical Engineering, University of Illinois. Development of 3D Structured Grid Generation Codes for Turbomachinery. May, 1992--Present.

David Modiano, Ph.D., Computational Fluid Dynamics, Massachusetts Institute of Technology, 1993. Application of Adaptive Mesh Refinement to Improve the Accuracy of Heat Transfer Calculations in Turbine Cooling Passages. February, 1993--Present.

Andrew Norris, Ph.D., Mechanical and Aerospace Engineering, Cornell University, 1993. Computation of Turbulent Reacting Flows by a Compressible Hybrid PDF Model. June, 1993--Present.

Robert Rubinstein, Ph.D., Mathematics, Massachusetts Institute of Technology, 1972. Theory and Modeling of Turbulence. May, 1993--Present.

Aamir Shabbir, Ph.D., Mechanical Engineering, State University of New York,Buffalo, 1987. Modeling of the Scalar Field of Turbulent Flow and Model Assessment. June, 1991--Present.

Shyue-Horng Shih, Ph.D., Aerospace Engineering, University of Cincinnati, 1993. Near-Field Calculation of Axisymmetric and 3D Supersonic Jet Noise. June, 1993--Present.

Tsan-Hsing Shih, Ph.D., Aerospace Engineering, Cornell University, 1984. Technical Leader, CMOTT Group. Developing, Validating and Implementing Improved Turbulence Models for Propulsion Systems. March, 1990--Present.

Erlendur Steinthorsson, Ph.D., Mechanical Engineering, Carnegie Mellon University, 1992. Code Development for Flows in Complex Geometries such as Turbine Blade Coolant Passages. January, 1992--Present.

Fu-Lin Tsung, Ph.D. expected 1994, Aerospace Engineering, Georgia Institute of Technology. Development of 3D Structured/Unstructured Hybrid Navier-Stokes Solver for Turbomachinery. March, 1993--Present.

J.J.W. van der Vegt, Ph.D., Mathematics, Delft University, Delft, The Netherlands, 1988. Development and Testing of Codes for the Direct Numerical Simulation of Transition and Turbulence in Compressible Boundary Layers. August, 1991--August, 1993.

Daniel Winterscheidt, Ph.D., Mechanical Engineering, University of Kansas, 1992. Finite Element Methods for Incompressible and Compressible Flows Involving Fluid-Structure Interactions. September, 1992--Present.

Zhigang Yang, Ph.D., Mechanical and Aerospace Engineering, Cornell University, 1989. Modeling of Bypass Transition and Stability Analysis of Swirling Flows. July, 1990--Present.

Shaye Yungster, Ph.D., Aeronautics and Astronautics, University of Washington, 1989. Development of CFD Codes for High Speed Combustion and Detonation Waves. November, 1989--Present.

Jiang Zhu, Ph.D., Mechanics, Institut National Polytechnique de Grenoble, 1986. Development of Efficient, Robust Codes for Propulsion System Flows that Can Accomodate Various Turbulence Models. April, 1992--Present.

B. Visiting Staff/Consultants.

Dare Afolabi, Ph.D., Mechanical Engineering, Imperial College, London, 1983. Associate Professor, Department of Mechanical Engineering, Purdue University, Indianapolis.

Andrea Arnone, Ph.D., Computational Fluid Dynamics, University of Bologna, Bologna, Italy, 1989. Assistant Professor, Department of Energy Engineering, University of Florence, Florence, Italy.

Thomas F. Balsa, Ph.D., Aerospace and Mechanical Engineering, Princeton University, 1970. Professor, Department of Aerospace and Mechanical Engineering, University of Arizona.

Giles J. Brereton, Ph.D., Mechanical Engineering, Stanford University, 1987. Assistant Professor, Department of Mechanical Engineering and Applied Mechanics, University of Michigan.

J. Y. Chen, Ph.D., Mechanical and Aerospace Engineering, Cornell University, 1985. Associate Professor, Department of Mechanical Engineering, University of California, Berkeley.

Frederic Coquel, Ph.D., Numerical Analysis, Ecole Polytechnique, Paris, France, 1992. Research Engineer, Theoretical Aerodynamics Branch I, ONERA, Chatillon, France.

Thong Dang, Ph.D., Aeronautics and Astronautics, Massachusetts Institute of Technology, 1985. Associate Professor, Department of Mechanical, Aerospace and Manufacturing Engineering, Syracuse University.

Peter W. Duck, Ph.D., Fluid Mechanics, University of Southampton, Southampton, United Kingdom. Reader, Department of Mathematics, University of Manchester, Manchester, United Kingdom.

Jack R. Edwards, Ph.D., Aerospace Engineering, North Carolina Agricultural and Technical State University, 1993. Research Associate, NASA Center of Research Excellence, North Carolina A&T State University.

Jitesh S. B. Gajjar, Ph.D., Mathematics, Imperial College,London, 1984. Lecturer, Department of Applied Mathematics, University of Manchester, Manchester, United Kingdom.

Max D. Gunzburger, Ph.D., Mathematics, New York University, 1969. Professor, Department of Mathematics and Interdisciplinary Center for Applied Mathematics, Virginia Polytechnic Institute and State University.

Thomas Hagstrom, Ph.D., Applied Mathematics, California Institute of Technology, 1983. Associate Professor, Department of Mathematics and Statistics, University of New Mexico.

S. I. Hariharan, Ph.D., Applied Mathematics, Carnegie Mellon University, 1980. Professor, Department of Mathematical Sciences, University of Akron.

George Huang, Ph.D., Mechanical Engineering, University of Manchester, Manchester, United Kingdom, 1986. Research Scientist, Eloret Institute, Palo Alto, California.

Arne Johansson, Ph.D., Department of Mechanics, The Royal Institute of Sweden, Stockholm, Sweden, 1983. Professor and Chairman, Department of Mechanics, The Royal Institute of Sweden, Stockholm, Sweden.

Prem K. Khosla, Ph.D., Mathematics and Sciences, Punjab University, Chandigarh, India, 1965. Professor, Department of Aerospace Engineering and Engineering Mechanics, University of Cincinnati.

Brian P. Leonard, Ph.D., Aerospace Engineering, Cornell University, 1965. Professor, Department of Mechanical Engineering, University of Akron.

Chaoqun Liu, Ph.D., Applied Mathematics, University of Colorado, Denver, 1989. Scientist, Front Range Scientific Computations, Inc. and Associate Professor (adjunct), University of Colorado, Denver.

Anastasios S. Lyrintzis, Ph.D., Aerospace Engineering, Cornell University, 1988. Assistant Professor, Department of Aerospace Engineering and Mechanics, University of Minnesota.

Sherwin A. Maslowe, Ph.D., Fluid Mechanics, University of California, Los Angeles, 1970. Professor, Department of Mathematics and Statistics, McGill University, Montreal, Canada.

Arthur F. Messiter, Ph.D., Aerodynamics and Mathematics, California Institute of Technology, 1957. Professor, Department of Aerospace Engineering, University of Michigan.

Roy A. Nicolaides, Ph.D., Computer Science, University of London, 1972. Professor, Department of Mathematics, Carnegie Mellon University.

Alexander Oron, Ph.D., Mechanical Engineering, Technion-Israel Institute of Technology, 1986. Senior Lecturer, Department of Mechanical Engineering, Technion-Israel Institute of Technology.

Christophe Pierre, Ph.D., Mechanical Engineering, Duke University, 1985. Associate Professor, Department of Mechanical Engineering and Applied Mechanics, University of Michigan.

Richard H. Pletcher, Ph.D., Mechanical Engineering, Cornell University, 1966. Professor, Department of Mechanical Engineering, Iowa State University.

Stanley G. Rubin, Ph.D., Aerospace Engineering, Cornell University, 1963. Professor, Department of Aerospace Engineering and Engineering Mechanics, University of Cincinnati.

William W. Schultz, Ph.D., Applied Mathematics and Engineering Mechanics, Northwestern University, 1982. Associate Professor, Department of Mechanical Engineering and Applied Mechanics, University of Michigan.

James N. Scott, Ph.D., Aeronautical and Astronautical Engineering, Ohio State University, 1977. Associate Professor, Department of Aeronautical and Astronautical Engineering, Ohio State University.

Wei Shyy, Ph.D., Mechanical Engineering, University of Michigan, 1982. Professor, Department of Aerospace Engineering, University of Florida.

J. Trevor Stuart, Ph.D., and Fellow, Royal Society Of London. Professor, Department of Applied Mathematics, Imperial College, London, United Kingdom.

Timothy W. Swafford, Ph.D., General Engineering, Mississippi State University, 1983. Associate Professor, Engineering Research Center for Computational Field Simulation, Mississippi State University.

Gretar Tryggvason, Ph.D., Engineering, Brown University, 1985. Associate Professor, Department of Mechanical Engineering and Applied Mechanics, University of Michigan.

Eli Turkel, Ph.D., Applied Mathematics, New York University, 1970. Professor, Department of Mathematics, Tel Aviv University, Tel Aviv, Israel.

Yasuhiro Wada, Ph.D., Aeronautics, University of Tokyo, 1992. Researcher, Computational Sciences Division, National Aerospace Laboratory, Tokyo, Japan.

David L. Whitfield, Ph.D., Aerospace Engineering, University of Tennessee, 1971. Professor and Director, Computational Fluid Dynamics Laboratory, Mississippi State University.

C. Graduate Students.

David Brown, Department of Aerospace Engineering and Engineering Mechanics, University of Cincinnati.

Scott A. Dudek, Department of Mechanical Engineering, University of California, Berkeley.

David R. Watkins, Department of Electrical Engineering, University of Akron.

	1986	1987	1988	1989	1990	1991	1992	1993
RESEARCHERS	23	43	50	46	47	49	58	64
SEMINARS	10	27	39	30	37	26	32	46
REPORTS	2	9	22	32	25	29	27	51
WORKSHOPS/LECT. SERIES	1	0	2	1	1	1	1	1
NO. OF PRESENTATIONS	7	0	21	14	15	21	15	33

TABLE II. - ICOMP STATISTICS (1986 TO 1993)

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Figure 1. ICOMP Visiting Researchers and Administrative Staff, July,1993. From bottom to top: Left Column; Theo Keith, S.H. Shih, Taiwit Chitsomboon, S.I. Hariharan, Thomas Balsa, David Modiano, Zhigang Yang, T.H. Shih, Shaye Yungster, Kumud Ajmani, J.K. Chung, Jiang Zhu. Center Column; Fu-Lin Tsung, Alexander Oron, Richard Pletcher, W.W. Liou, Daniel Winterscheidt, Chaoqun Liu, D.R. Reddy,(NASA Lewis), Robert Deissler, James Loellbach. Right Column; Karen Balog, Louis Povinelli, Charles Feiler, Giles Brereton, Aamir Shabbir, Kai Kao, Lin-Jun Hou, Bo-Nan Jiang, Frederic Coquel, Andrew Norris, Erlendur Steinthorsson, Eli Reshotko(CWRU).

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)

2. REPORT DATE
April 1994

3. REPORT TYPE AND DATES COVERED
Technical Memorandum

4. TITLE AND SUBTITLE
Institute for Computational Mechanics in Propulsion (ICOMP)

5. FUNDING NUMBERS
WU-505-90-5K

6. AUTHOR(S)
Charles E. Feiler, editor

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135-3191

8. PERFORMING ORGANIZATION REPORT NUMBER
E-8686

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, D.C. 20546-0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA TM-106542
ICOMP-94-01

11. SUPPLEMENTARY NOTES
Report compiled and edited by Dr. Charles E. Feiler, ICOMP Executive Officer, and approved by Louis A. Povinelli, ICOMP Director, NASA Lewis Research Center (work funded under NASA Cooperative Agreement NCC3-233). ICOMP Program Director, Louis A. Povinelli, organization code 2600, (216) 433-5818.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Categories 34 and 64

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by the Ohio Aerospace Institute (OAI) and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the accomplishments and activities at ICOMP during 1993.

14. SUBJECT TERMS
Numerical analysis; Computer science; Mathematics; Fluid mechanics

15. NUMBER OF PAGES
91

16. PRICE CODE
A05

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	